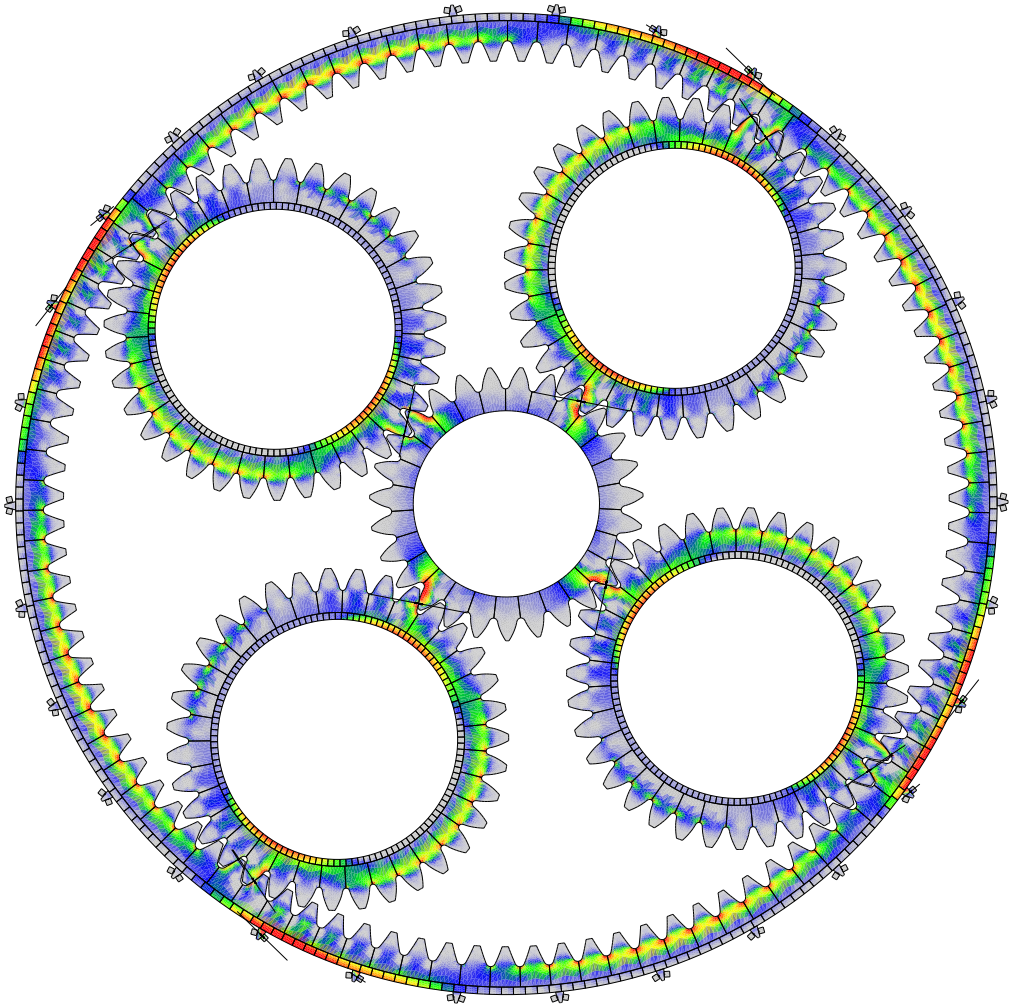
Planetary2D User's Manual



Advanced Numerical Solutions
Hilliard OH

April 25, 2006

Contents

Preface **xiii**

1 Introduction **1**

2 Planetary2D Software Package **5**
2.1 Planetary2D analysis package 5
2.2 Installation of the software package on windows platform 8

3 Preliminaries **9**
3.1 System of units 9
3.2 Bodies 9
3.3 Reference frames 9
3.4 Bearings 13
3.5 Contact solver, contact modelling and grid specification 15
3.6 The main menu 15

4 The Graphical User Interface **17**
4.1 Menu command items 17
4.2 Integer menu items 19
4.3 Floating point menu items 19
4.4 Boolean menu items 19
4.5 String menu items 19
4.6 Switch type menu items 20
4.7 Commonly occurring buttons 20
4.8 Graphics 21

5 Building a Model **23**
5.1 System level data 24
5.2 System sub-menus 28
5.3 Pinion data 32
5.4 Bearings 32
5.5 Tooth numbering 35
5.6 Spacing error menu 36
5.7 Thickness errors 37
5.8 Modelling the tooth profile 39
5.9 Rayleigh Damping Model 43
5.9.1 Modal Damping Ratio - SDOF system 43
5.9.2 Modal Damping Ratio Measurement 44
5.10 Surface modifications 46
5.11 Modelling the rim 50
5.12 Sun gear data 52

5.13 Sun gear bearings . . . 52
5.14 Ring data . . . 54
5.15 Carrier data . . . 58

6 Running an Analysis 61
6.1 Surface gages . . . 61
6.2 Finite element probes . . . 62
6.3 Load sensors . . . 65
6.4 The setup menu . . . 65
6.5 Other output files . . . 66

7 Pre- and Post-processing 69
7.1 Selecting bodies . . . 71
7.2 View parameters . . . 71
7.3 The DRAWBODIES command . . . 75
7.4 The NUMBER command . . . 78
7.5 The TOOTHLOAD command . . . 78
7.6 The CONTACT command . . . 82
7.7 The TOOTHLDHIST command . . . 84
7.8 The SUBSURFACE command . . . 84
7.9 The GRIDLDHIST command . . . 87
7.10 The GRIDPRHIST command . . . 87
7.11 The SEPBEFHIST command . . . 91
7.12 The SEPAFTHIST command . . . 91
7.13 The SEARCHSTRESS command . . . 95
7.14 The POINTSTRESS command . . . 95
7.15 The PATTERN command . . . 101
7.16 The AUDIT command . . . 102
7.17 The BODYDEFLECTION command . . . 104
7.18 The BODYREACTION command . . . 104
7.19 The BRGDEFORMN command . . . 108
7.20 The BRGREACTION command . . . 108

8 Pre and Post processing using IglassViewer 113
8.1 Generating an Iglass file for preprocessing . . . 113
8.2 View menu . . . 116
8.2.1 Finite element mesh . . . 116
8.2.2 Cutting plane . . . 116
8.2.3 Selecting the time step . . . 116
8.2.4 Reference frames . . . 119
8.3 The Bodies menu . . . 119
8.4 Post processing using iglass . . . 120
8.5 Features specific to iglass post processing . . . 120

9 Running a dynamic case 127
9.1 The EDIT menu . . . 127
9.2 The System level data . . . 127
9.3 The Angular position data . . . 127
9.4 The Pinion data . . . 127
9.5 The Pinion tooth modification . . . 129
9.6 The Pinion bearing data . . . 129
9.7 The Pinion Rim data . . . 129

9.8 The Sun gear data 130
9.9 The Sun tooth modification 130
9.10 The Ring gear data 130
9.11 The Ring gear tooth modification 132
9.12 The Spline data for the Ring gear 132
9.13 The Ring gear rim data 132
9.14 The Carrier data 132
9.15 Calculating the time step for dynamic problems 134
9.16 The setup menu 135
9.17 Analysis and Results 137
9.17.1 Calculating the Dynamic load factor 141
9.18 Applying the 2D Planetary program for modal analysis 144

A Tooth Mesh Templates **155**

List of Figures

1.1 The 2D planetary gear set 2

2.1 The computer programs in the *Planetary2D* analysis package 6
2.2 The menu presented to the user by Guide 7

3.1 A multi-body system 10
3.2 Reference frame degrees of freedom 11
3.3 The reference frames set up for a 2D planetary gear set, shown at time $t = 0$ . . 12
3.4 Bearing connections in the multi-body model. 13
3.5 Bearing races 13
3.6 Bearing deformation 14
3.7 Bearing reaction 14
3.8 The main menu. 16

4.1 Planetary2D user interface 18
4.2 An integer data entry box 19
4.3 An floating point data entry box 19
4.4 An boolean data entry box 19
4.5 An string data entry box 19
4.6 An switch type data entry box 20

5.1 The EDIT menu. 23
5.2 The system data menu. 25
5.3 Sun and ring gear assembly errors. 27
5.4 The pinion assembly errors. 29
5.5 The Pinionrunout menu. 30
5.6 The Angposnpinion menu. 30
5.7 The Pinposnerror menu. 31
5.8 The Pinion main menu. 32
5.9 The Bearing menu. 34
5.10 The tooth and side numbering scheme for the pinion and sun gear 35
5.11 The tooth and side numbering scheme for the ring gear 35
5.12 The Spacing error menu. 36
5.13 The Spacing error submenu. 36
5.14 The Thickness error menu. 37
5.15 The tooth data menu for the pinion gear. 39
5.16 Diameter definitions for an external gear. 40
5.17 Frequency Response of a SDOF system. 44
5.18 The tooth modification menu. 46
5.19 Linear tip modification 47

5.20 Quadratic tip modification 48
5.21 Tabular tip modification 49
5.22 The menu for specifying rim data for pinion. 50
5.23 The Sun gear menu. 52
5.24 The Sun bearing menu. 53
5.25 The Ring data menu. 54
5.26 The Spline data menu. 55
5.27 Ring gear spline geometry 56
5.28 The carrier menu. 58

6.1 The surface gage menu 63
6.2 The finite element probe menu 64
6.3 The load sensor menu 65
6.4 The setup menu 67

7.1 The pre-processing menu. 69
7.2 The post-processing file name dialog box. 70
7.3 The post-processing menu. 70
7.4 The body selection menu. 71
7.5 The view menu in pre-processing mode 72
7.6 The view menu in post-processing mode. 73
7.7 The view menu in post-processing mode with the LOADS option enabled. 74
7.8 An example of a drawing made in the post-processing mode. 75
7.9 An example of a drawing made in the post-processing mode. 76
7.10 An example of a drawing made in the post-processing mode using the exaggeration command. 77
7.11 The NUMBER menu. 78
7.12 Tooth numbering superimposed on a sun gear drawing, using the NUMBER command. 79
7.13 The TOOTHLOAD menu. 80
7.14 The toothload vs. time graph generated by the TOOTHLOAD menu 81
7.15 The CONTACT menu. 82
7.16 The tooth contact pressure vs. time graph generated by the CONTACT menu. . 83
7.17 The TOOTHLDHIST menu. 84
7.18 The tooth load histogram generated by the TOOTHLDHIST menu. 85
7.19 The SUBSURFACE menu. 86
7.20 The GRIDLDHIST menu. 87
7.21 The gridload histogram generated by the GRIDLDHIST menu. 88
7.22 The GRIDPRHIST menu. 89
7.23 The gridload histogram generated by the GRIDPRHIST menu. 90
7.24 The SEPBEFHIST menu. 91
7.25 The histogram of grid separation before contact, generated by the SEPBEFHIST menu. 92
7.26 The SEPAFTHIST menu. 93
7.27 The histogram of grid separation after contact, generated by the SEPAFTHIST menu. 94
7.28 The SEARCHSTRESS menu 96
7.29 The graph of root stress vs. time, generated by the SEARCHSTRESS menu. . . 97
7.30 The graph of root stress vs. profile, generated by the SEARCHSTRESS menu. . 98
7.31 The POINTSTRESS menu 99
7.32 The graph of root stress vs. time, generated by the POINTSTRESS menu. . . . 100
7.33 The PATTERN menu. 101

7.34 The AUDIT menu. . . . 103
7.35 The BODYDEFLECTION menu. . . . 104
7.36 The graph generated by the BODYDEFLECTION menu. . . . 105
7.37 The BODYREACTION menu. . . . 106
7.38 The graph generated by the BODYREACTION menu. . . . 107
7.39 The BRGDEFORMN menu. . . . 108
7.40 The graph generated by the BRGDEFORMN menu. . . . 109
7.41 The BRGREACTION menu. . . . 110
7.42 The graph generated by the BRGREACTION menu. . . . 111

8.1 The generate Iglass file menu . . . 114
8.2 An example of an Iglass preprocessing window. . . . 115
8.3 Iglass preprocessing view menu . . . 116
8.4 Finite element mesh model of the gear bodies . . . 118
8.5 The cutting plane switch. . . . 118
8.6 The position slider. . . . 118
8.7 The time menu. . . . 118
8.8 The reference frame switch. . . . 119
8.9 Iglass preprocessing Bodies menu . . . 119
8.10 The generate iglass file menu for post processing. . . . 120
8.11 An example of an iglass post processing window. . . . 121
8.12 The position slider. . . . 122
8.13 The deformation slider. . . . 122
8.14 The load slider. . . . 122
8.15 The bearing forces and moments sliders. . . . 122
8.16 The iglass postprocessing attribute menu. . . . 123
8.17 The attribute switch. . . . 123
8.18 The palette switch. . . . 123
8.19 Finite element mesh so as to find the stress at a nodal point . . . 124
8.20 The background color popup window switch. . . . 124
8.21 The Contact pattern menu. . . . 125

9.1 Graph showing the dynamic maximum principal normal stresses (s1) and minimum principal normal stresses (s3) against time for the fillet surface of the sun gear, measured by Surfage Gage 1. The results are obtained from the file GAGES.DAT. . . . 137
9.2 Graph showing the pinion bearing tangential displacement against time for dynamic analysis. The tangential displacement at each instant of time is given in Column no.3 of the PINION1BRGRES.DAT file. . . . 139
9.3 Graph showing the housing moments against time for dynamic analysis. The moment about the Z-axis for each time instant is given in column no.13 of the HOUSINGRES.DAT file . . . 140
9.4 Graph showing the dynamic tooth load against time. The plot is obtained using the load values from column nos.6, 10, 11 and 12 from the LOADS.DAT file. . . . 141
9.5 Graph showing the static tooth load against time using the TOOTHLOAD command. . . . 143
9.6 Plot showing the radial component frequency response of pinion1 in the planetary system in the dynamic state (transient response is in column no.2 of the PINION1BRGRES.DAT file) . . . 146
9.7 Plot showing a narrow range of radial component frequency response of pinion1 in the planetary system in the dynamic state (transient response is in column no.2 of the PINION1BRGRES.DAT file) . . . 147

9.8 Plot showing the response near the first multiple of mesh frequency for the radial component of deformation of the pinion1 bearing (transient response is in column no.2 of the PINION1BRGRES.DAT file) . . . 148
9.9 Plot showing the tangential component frequency response of pinion1 in the planetary system in the dynamic state (transient response is in column no.3 of the PINION1BRGRES.DAT file) . . . 149
9.10 Plot showing a narrow range of tangential component frequency response of pinion1 in the planetary system in the dynamic state (transient response is in column no.2 of the PINION1BRGRES.DAT file) . . . 150
9.11 Plot showing the response near the second multiple of the mesh frequency for the tangential component of deformation of the pinion1 bearing (transient response is in column no.2 of the PINION1BRGRES.DAT file) . . . 151
9.12 Plot showing the mode shape at pinion1 in the planetary system in the dynamic state at 574.05Hz . . . 152
9.13 Plot showing the mode shape at the sun gear in the planetary system in the dynamic state at 574.05Hz . . . 153

A.1 The MEDIUM.TPL template file. . . . 156
A.2 The FINEROOT.TPL template file. . . . 157
A.3 The FINEST.TPL template file. . . . 158
A.4 The THINRIM.TPL template file. . . . 159

List of Tables

4.1 Common buttons 20

5.1 System configuration parameters 26
5.2 Pinion runout error menu 28
5.3 Pinion angular position menu 28
5.4 Pin position error menu 30
5.5 Pinion lumped parameters 33
5.6 Bearing menu parameters 33
5.7 Spacing error menu 37
5.8 Thickness error menu 38
5.9 Parameters describing the tooth profile for pinion 41
5.10 Rayleigh damping parameters for steel. 43
5.11 Linear tip modification parameters 47
5.12 Quadratic tip modification parameters 47
5.13 Tabular profile modification parameters 48
5.14 Rim data parameters 51
5.15 Spline data parameters 57
5.16 Carrier data parameters 59

8.1 Common buttons in Iglass pre and postprocessing window 117

9.1 Physical units 128
9.2 System level data 128
9.3 Pinion angular position data 128
9.4 Data describing the tooth profile for pinion 129
9.5 Quadratic tip modification data 129
9.6 Bearing menu parameters 129
9.7 Pinion Rim data 130
9.8 Data describing the tooth profile for sun gear 130
9.9 Tabular profile modification data for the Sun gear tooth 131
9.10 Data describing the tooth profile for Ring gear 131
9.11 Tabular profile modification data for the Ring gear tooth 132
9.12 Spline data for the Ring gear 132
9.13 Rim data parameters for the ring gear 133
9.14 Carrier data 133
9.15 Setup menu 136
9.16 Surface gage data for extracting the plot shown in Figure 9.1 138
9.17 Load sensor data for extracting the plot shown in Figure 9.4 142
9.18 Tooth load data for extracting the plot shown in Figure 9.5 142

9.19 Displacement values in the radial direction for a critical frequency of 574.05 Hz for the pinions 146
9.20 Displacement values in the tangential direction for a critical frequency of 574.05 Hz for the pinions 147

Preface

The Planetary2D computer program has been under development for many years, and is finally available for use by the gearing community. We have received active support and encouragement from many people. We would especially like to thank Timothy Krantz of the Army Research Laboratory at the NASA Glenn Research Center for his support and encouragement.

Sandeep Vijayakar, Hilliard OH
Samir Abad, Hilliard OH
February 2003

Chapter 1

Introduction

Planetary gear systems yield several advantages over conventional parallel shaft gear systems. They produce high speed reductions in compact spaces, greater load sharing, higher torque to weight ratio, diminished bearing loads and reduced noise and vibration. They are used in automobiles, helicopters, aircraft engines, heavy machinery and a variety of other applications. Despite their advantages, noise induced by the vibration of planetary gear system remains a key concern. In helicopter, for example, cabin noise exceeding $100db$ is directly traceable to the last stage planetary gear mounted to the cabin.

Dynamic response calculations of Planetary gears have received considerably less research attention than single mesh gear pairs. The purpose of this work is to demonstrate the application of *Planetary2D* program for static and dynamic problems. The analytical technique used to design the software package combines a unique, semi-analytical finite element approach with detailed contact modeling at the tooth mesh. This approach was specifically developed to examine the mechanics of precisely machined, contacting elastic bodies such as gears. The semi-analytical finite element approach does not require a highly refined mesh at the contacting tooth surfaces. This dramatically reduces the computational effort and allows calculation of the dynamic response at a sufficient number of time steps to study the response in the frequency domain. In contrast, the need for extremely refined gear tooth meshes limits conventional finite element analysis to static analysis and free body eigen solutions. Experimental measurement of planetary gear dynamic response under operating conditions is difficult due to limited access and multiple moving bodies. The *Planetary2D* analysis package provides a more accurate and comprehensive study of planetary gear dynamic response than is reasonably possible, or has been conducted, with conventional finite element analysis. A typical *Planetary2D* gear set is shown in Figure 1.1.

Contact pair analysis of a planetary gear set is difficult for general purpose finite element software for many reasons.

- *Size of the contact zone:* The width of the contact zone in typical gearing applications is two orders of magnitude smaller than the dimensions of the gear teeth themselves. In order to model the contact conditions with sufficient accuracy, a general purpose non-linear finite element program needs to have a large number of nodes (a very fine mesh) inside the contact zone. To run such a contact model the fine mesh in the contact zone has to transition into a much coarser mesh over the rest of the gear. The location of the contact zone, however, changes as the gears move. This means that either the gear finite element model should be re-meshed for each time instant, or that the finite element mesh be highly refined over its entire surface area. Both these alternatives lead to unacceptably high computational costs. Our approach has been to use the finite element models only to compute relative deformation and stresses for points that are away from the contact

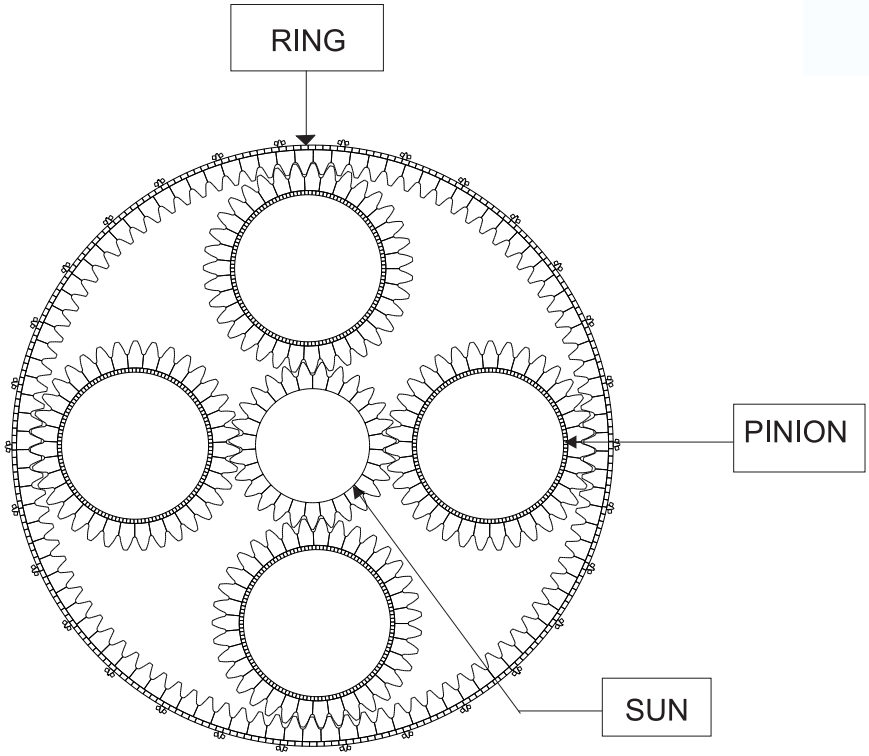


Figure 1.1: The 2D planetary gear set

zones. For points within the contact zone, we use semi-analytical techniques to compute the relative deformations and stresses. The 'near field semi-analytical solution and the 'far field finite element solutions are matched at a 'matching surface. Such a model is significantly difficult to program on a computer, but once implemented, can provide much better resolution without using a highly refined finite element mesh.

- *Rigid body degrees of freedom in the planetary system:* In multi-mesh gear systems like planetary transmissions, there are many rigid body degrees of freedom or mechanisms that are constrained only by the contact conditions. This means that if a non-linear finite element code with *gap elements* is used, then the incremental stiffness matrices become singular. Most commercial codes don't work when this happens. Some manupalations are commonly used, such as adding imaginary linear and torsional springs to make the system stiffness matrices non-singular. The spring stiffness can be made small, but the accuracy of results computed by such almost singular stiffness matrices is questionable. Our approach has been to attach a reference frame to each individual component, and to carry out the finite element computations for each individual component separately in its own reference frame. As long as each finite element mesh is sufficiently well constrained to its reference frame, the stiffness matrices are well behaved. The free mechanisms in the system can be modeled by allowing the reference frames to move freely. The contact solver used is based on the *Revised Simplex Solver*. This Solver is commonly used to solve quadratic programming problems. It can take into account any free mechanisms in the system while computing the contact loads.

- *System Kinematics:* The nominal position of each individual gear in the system changes with time. The nominal positions of the components are determined by the kinematics of the system. The kinematics of the system affects the nominal sliding velocities and inertial loads. It is very difficult to include this kinematic information into the finite element programs currently available. We have built a special purpose programming language into the software in order to specify the details of the kinematics of each component in the system. Important details such as the kinematic effect of assembly errors, runout and misalignments are easy to apply using this approach.

- *Large number of degrees of freedom:* For a system in which the total number of gear teeth is about 200, the total number of finite element degrees of freedom can be extremely large. This is so even with the finite element model refined only as much as is necessary for the far field solution. The total number of finite element degrees of freedom is approximately proportional to the total number of teeth. The amount of CPU time and memory needed to run a dynamic analysis with such a large degree of freedom would make it impractical. We have resorted to using a hierarchical representation of the system, in which the system is built from many substructures, with each substructure in turn being composed of many substructures. The processes of stiffness decomposition and load vector back-substitution now become very complex, and involve multiple recursive traversals of the substructure hierarchy. However, it is now possible to keep CPU requirements within practical limits.

- *Convergence of conditions at contact interfaces:* Poor convergence of contact conditions at interfaces is one of the biggest problems caused by using a general non-linear solver to solve a problem with contact constraints. The constraints imposed by the contact between mating surfaces are essentially linear inequality constraints. When a general-purpose non-linear solver is used to solve this problem, convergence is not guaranteed, and if it does occur, it is usually very slow. The *Revised Simplex solver* that we use provides a guarantee of convergence within a predetermined number of iterations. Furthermore, ill-posed contact problems can be detected even before the solution process is started. The solver is specifically designed for the linear inequality type constraints found in contact problems.

Chapter 2

Planetary2D Software Package

This chapter explains the various features of the *Planetary2D* software package.

2.1 Planetary2D analysis package

Calyx is a powerful contact analysis code capable of analyzing a variety of contact problems, including 2D and 3D dynamic and static analysis of systems such as gears, compressors, and brakes. Because *Calyx* has to be capable of handling a variety of problems, it communicates with the outside world through a programming language. The programming language interface of *Calyx* brings flexibility at the expense of ease of use. Such an interaction is appropriate for an advanced *Calyx* user, but not for a gear design engineer.

In order to address this issue, the program *Multyx* is used. *Multyx* is capable of communicating with the user through an easy to use menu-based interface. It translates the user's commands into the appropriate programming language statements and sends them on to *Calyx*. A typical user does not even need to know that *Calyx* is running in the background.

In addition to the user interface, Multyx also has built-in model generators. The Planetary 2D models described in this manual are all generated by *Multyx*. It also has post-processing and data extraction code, to help the user extract the results of analysis from *Calyx*.

Multyx and *Calyx* are designed as portable code, and can run on any system that supports standard C++. In order to keep it portable, *Multyx*'s menu system is command line based, and does not use any of the GUI features such as buttons, windows or mouse interaction. The following dialog shows some of the command line interface of *Multyx*.

```
E:>multyx
MultyX v.1.06, Copyright Advanced Numerical Solutions Dec 21 2000
MultyX>post ok patt
MultyX.PostProc.1/11.Pattern>HELP
MENU            Show menu
?               Show menu
HELP            Show menu
EXIT            Return to main menu.
QUIT            Return to main menu.
START           Draw the contact pattern.
CLEAR           Clear the graphics page.
SURFACEPAIR     Surface pair (Currently=PINION1_SURFACE1_SUN_SURFACE1_)
MEMBER          Member (Currently=PINION1)
TOOTHBEGIN      20              Tooth no. or instance no. of surface.
TOOTHEND        2               Tooth no. or instance no. of surface.
```

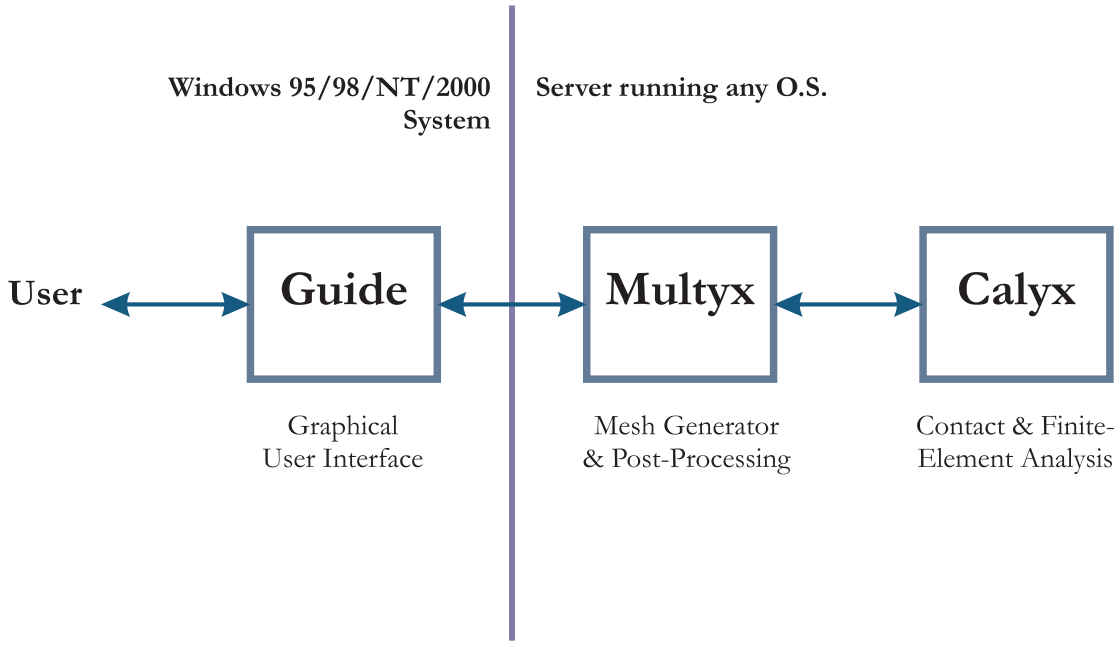


Figure 2.1: The computer programs in the *Planetary2D* analysis package

```
BEGINSTEP       1               Time/Roll angle step at which to begin search.
ENDSTEP         11              Time/Roll angle step at which to end search.
COLORS          Whether to render the model in color (Enabled)
CONTOURS        Whether to draw pressure contours (Enabled)
MINPRESS        4.000000E+004   Level of lowest press. contour.
MAXPRESS        4.200000E+005   Level of highest press. contour.
DELTAPRESS      4.000000E+004   Spacing between press. contours.
SMOOTH          (FALSE)         Whether to smooth the pressure contours.
OUTPUTTOFILE    Whether to write data to file. (Disabled)
MultyX.PostProc.1/20.Pattern>START
```

Guide is a program that provides a Graphical User Interface (GUI) to *Multyx*. *Guide* translates each of *Multyx*'s dialogs and presents them to the user in a graphical form. The command line menu described above is presented to the user as shown in Figure 2.2.

In addition, *Guide* provide the user with convenient ways of viewing the graphics, and helps the user convert the graphics into Microsoft formats and into Encapsulated PostScript (EPS) files.

Although *Guide* enhances the friendliness of *Multyx*, it is not required. All the features of *Multyx* can be accessed without *Guide*. The connection between *Guide* and Multyx is based on the TCP/IP telnet protocol when they are running on different computers. When running on the same computer, they communicate through named pipes. *Guide* is a heavy user of advanced operating system features including GUI support, multi-threading support, and inter-process communication support. *Guide* now runs on Windows 95/98/NT/2000 systems only.

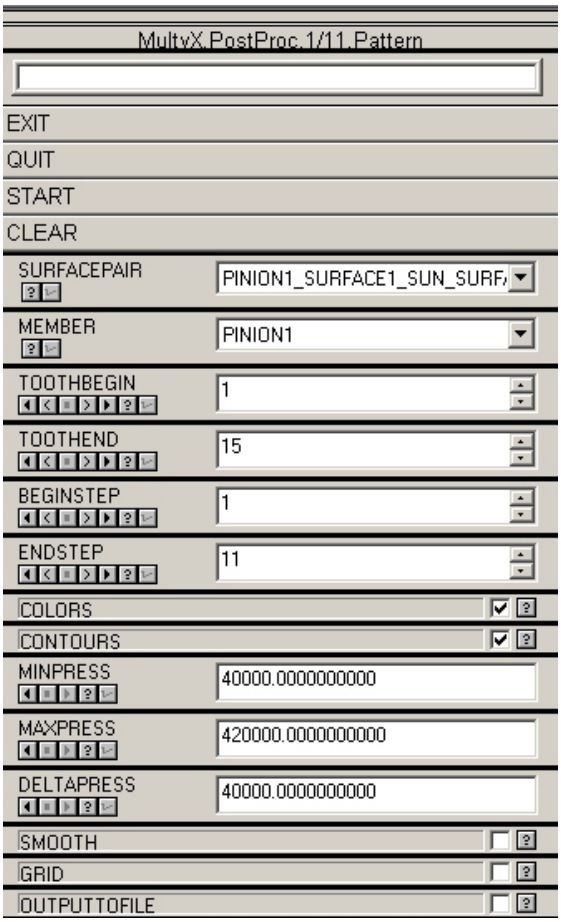


Figure 2.2: The menu presented to the user by Guide

2.2 Installation of the software package on windows platform

The procedure for installing the software analysis package on windows 95/98/NT platforms is as follows:

- Get the self-extracting file `setup.exe` from Ansol(Advanced numerical solutions) either by email, ftp or on a floppy. If you are using an ftp client, make sure that the `setup.exe` file is downloaded when ftp is in binary mode.
- Before installing anything make sure that there are no previous copies of `guide.exe`, `multyx.exe` or `calyx.exe` in your path. If these files are present then you either have to move the old programs elsewhere or change the path so that they don't conflict with the new programs.
- Now you are all set to install the software package. Close all the other programs and run `setup.exe`. It will ask you questions about where to install the program and where to keep the working directory.
- After you answer these questions, it will display the *Computer-ID* and ask for a *licence key*. Copy the *Computer-ID* and click the button -*skip(or install key later)*. It will proceed with the installation and will install 3 icons under *Start/Programs/2DPlanetary Analysis*.
- Send the *Computer-ID* via email to support@ansol.com, and we will send you the *Licence Key*. If you forget to copy the *Computer-ID*, run the icon- *Start/Programs/2DPlanetary Analysis/Register* and copy it again. Click on the *skip* button.
- After you receive the *Licence Key* from Ansol run- *Start/Programs/ 2DPlanetary Analysis/Register* again and paste the *Licence Key* in the respective box. Now, click on the button-*Install Licence Key*.
- Now you are all set to run the analysis. Start the program by using the icon *Start/Programs /2DPlanetary Analysis/2DPlanetary*.

Chapter 3

Preliminaries

The previous chapter gave an overview of the software architecture. This chapter provides some information to help you get up and running with the program.

3.1 System of units

Any system of units can be used provided that all the inputs provided by the user are in consistent with this system of units. The user is free to choose any units for force, time and length. All the inputs should then be in units that are consistent with this choice. For example, if the user chooses Kgf as the unit for force, seconds as the unit for time, and cm as the unit for length, then the input torque should be in $Kgf.cm$, the Youngs modulus in Kgf/cm^2, the Diametral pitch in $1/cm$ and the mass density in $Kgf.s^2/cm^4$. Outputs will also appear in consistent units.

3.2 Bodies

In multi-body contact analysis, the term 'body' is used to refer to an object that is capable of rigid body motion, and interacts with other bodies through surface contact and bearing connections.

There is a special body called the 'fixed body' which refers to ground. Figure 3.1 shows a typical multi body system.

In a 2D Planetary system the sun gear, pinions and the ring gear are all treated as separate bodies. The sun-pinion and the pinion-ring interaction is through contact. The *spider* or the *carrier* supports the planet shafts. The pinions revolve around the central axis while rotating along their own axes at the same time, much like planets revolve around the sun. The interaction between the carrier and the pinion shafts is also through contact.

3.3 Reference frames

Each of the bodies in the system has a reference frame to which it is rigidly attached. The reference frame has 6 rigid body type degrees of freedom, three translation components U_x, U_y and U_z, and three rotation components θ_x, $\theta_y$and and θ_z (Figure 3.2).

In addition to the body reference frames, there is a special reference frame called the fixed reference frame that is considered as 'ground', and does not move. It is used as the reference for defining the locations of all other reference frames.

Figure 3.3 shows how the pinion, sun, ring and carrier reference frames are set up relative to the fixed reference frame for a 2D planetary gear set at time $t = 0$. The fixed frame is located with its origin coinciding with the center of the carrier. The pinion, sun, ring and carrier

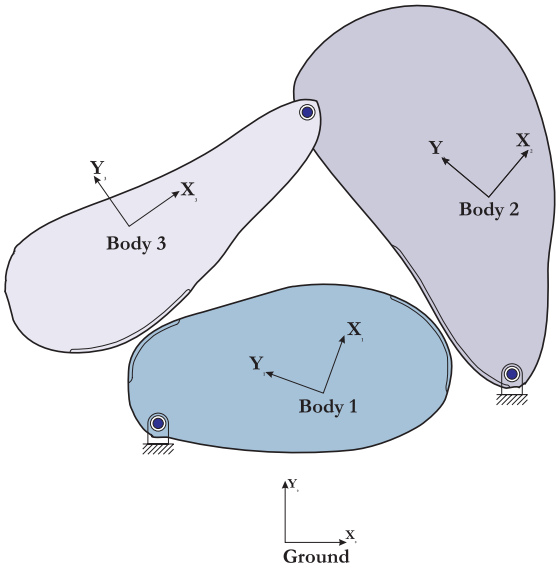


Figure 3.1: A multi-body system

reference frames have their origins at their centers of rotation. At time $t = 0$, the sun, ring and carrier X, Y axes are parallel to the corresponding axes of the fixed reference frame.

Manufacturing and assembly errors applied to the system might perturb the location of these reference frames slightly from their nominal location. In their nominal location, the origin of the sun and ring gears coincide with the origins of the fixed and carrier frames.

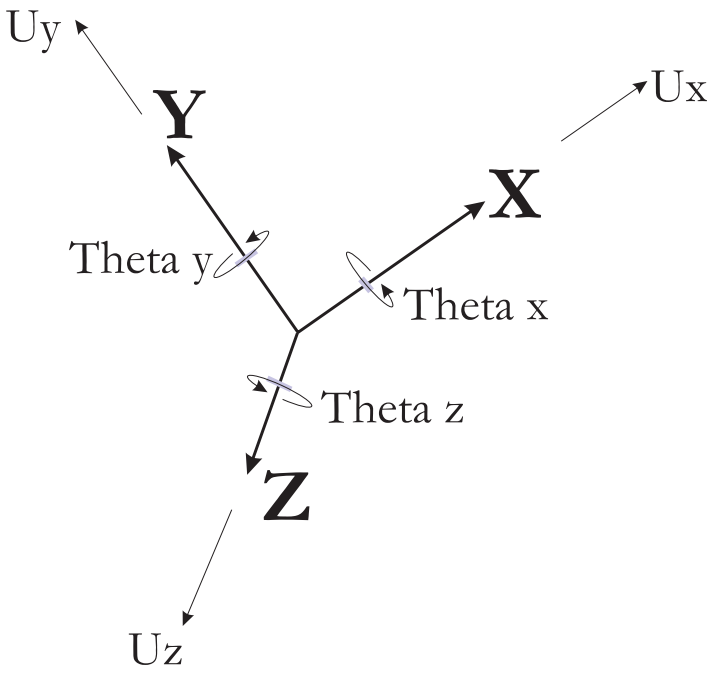


Figure 3.2: Reference frame degrees of freedom

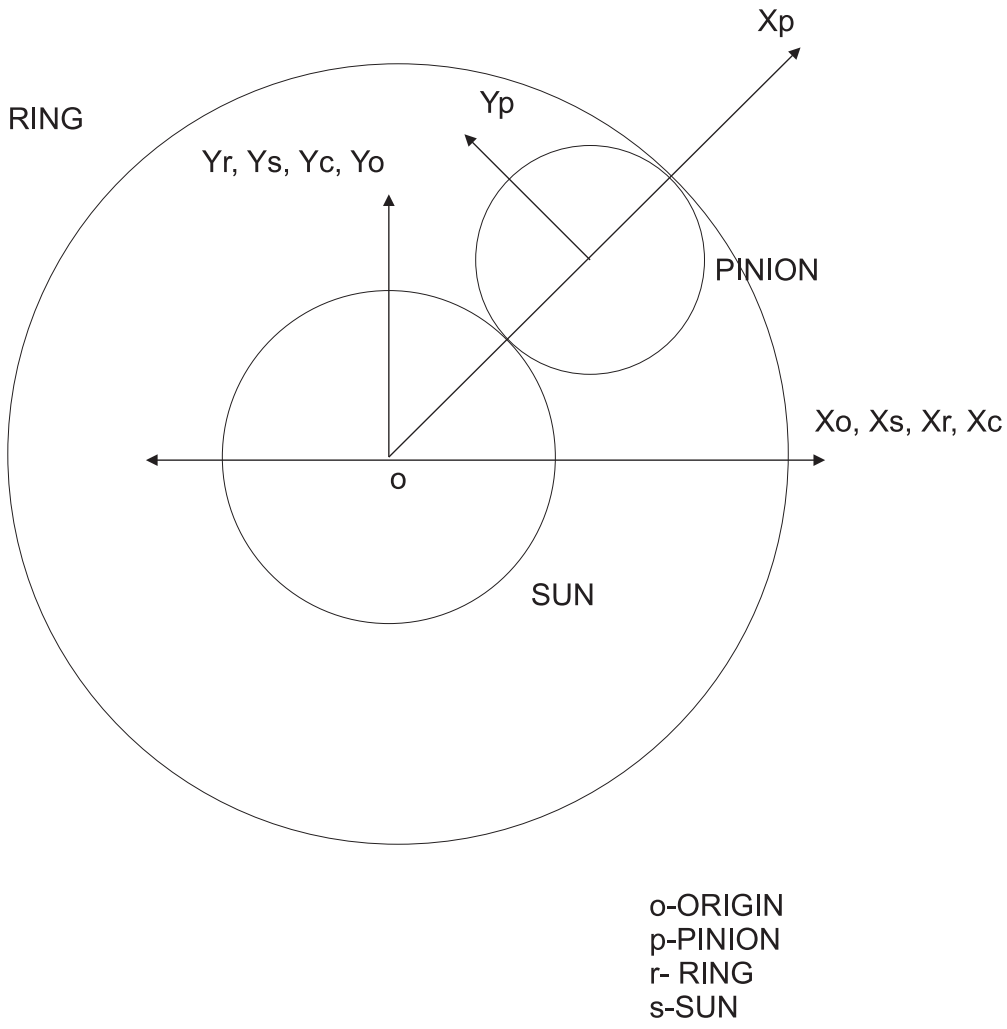


Figure 3.3: The reference frames set up for a 2D planetary gear set, shown at time $t = 0$

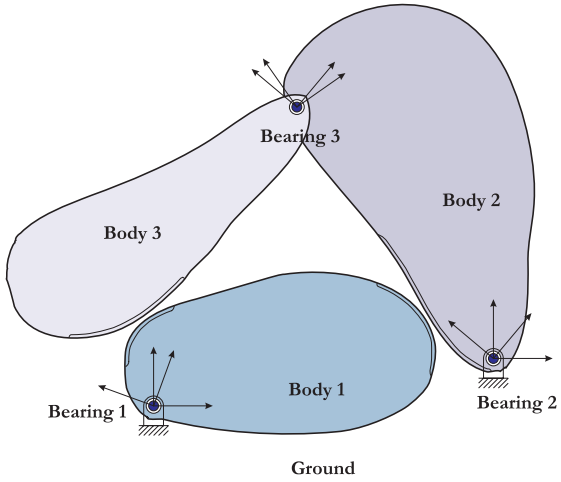


Figure 3.4: Bearing connections in the multi-body model.

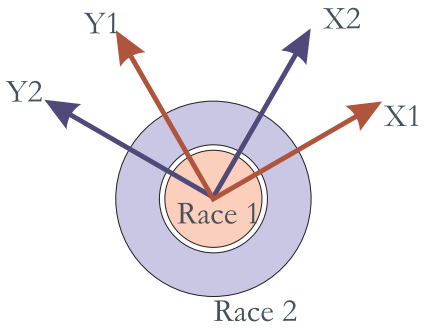


Figure 3.5: Bearing races

3.4 Bearings

In a multi-body system, bodies can also interact through bearings (Figure 3.4).

Bearings are treated as a stiffness connection between two bearing 'races' (Figure 3.5). Each race has an attached reference frame. The race is treated as a rigid body, and the six degrees of freedom of the first bearing race and the six degrees of freedom of the second bearing race are inter-related through a stiffness matrix.

The six degrees of freedom are the three translation degrees of freedom U_x, U_y and U_z, and three rotation degrees of freedom θ_x, $\theta_y$and and θ_z. The degrees of freedom represent the motion of race 1 relative to race 2. The components are measured in the reference frame attached to race 2 as shown in Figure 3.6.

The bearing may also generate internal reaction forces and moments. The six components of bearing reaction (Figure 3.7) consist of three forces F_x, F_y and F_z and the three moments M_x, M_y and M_z. Again, these reactions are those exerted by race 1 on race 2. The components are computed in the reference frame of race 2.

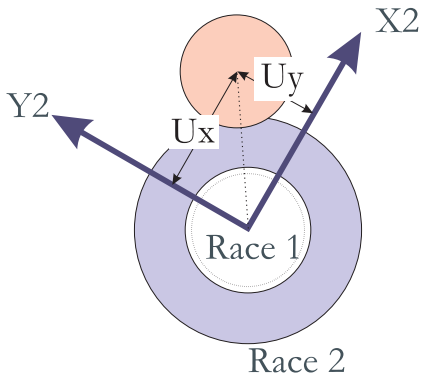


Figure 3.6: Bearing deformation

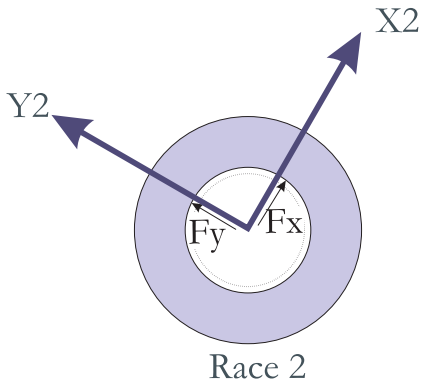


Figure 3.7: Bearing reaction

3.5 Contact solver, contact modelling and grid specification

At each time step, the unknown contact loads have to be determined by a Contact Algorithm. Calyx poses the contact problem in the form of a quadratic programming problem and then uses the Revised simplex algorithm to efficiently solve the unknown loads. The advantage of this technique is that convergence is excellent, and there is a mathematical proof of convergence.

Calyx carries out a search over all possible combinations of instances of all specified surface pairings to determine which surface instances could make contact. To eliminate very widely separated surface instances from further analysis, it uses a parameter called the separation tolerance, or SEPTOL in Multyx. The value of this parameter is specified by the user, and is in units of distance. If two surfaces are separated by more than this distance the pair of surfaces are discarded for further analysis.

Each two-dimensional surface is parameterized by a parameter s, which varies from 0 at one end of the surface to a maximum value, say n at the other end of the surface. During contact analysis, Calyx finds the point on the surface that is most likely to make contact, and then lays a contact grid around the point. The grid is placed at equal intervals of s away from the most likely point of contact. This width of each grid 'cell' is specified by the user. It is called DSPROF in Multyx. The number of such "patches" laid out on either side of the most likely point of contact can also be specified in the configuration file, and is called NPROFDIVS in Multyx.

3.6 The main menu

The *Planetary2D* package is started by clicking on an icon created during the installation process. After the *Planetary2D* package is started, the main menu shown in Figure 3.8 comes up.

All user provided data is saved in a file called the session file. The name of this session files can be changed by typing the name in the SESFILENAME box. Changing the file name does not actually write the data to the new file, nor does it read data from the new file. Data is written to the session file through the SAVESESSION command. Data can be loaded from an existing session file using the LOADSESSION command.

The QUIT command terminates the program without saving any data in the session file. The EXIT command first writes data to the session file, and then terminates the program.

All data entry occurs in a hierarchy of submenus accessed through the EDIT command on this main menu.

After data entry is completed, the GENERATE command may be used to generate the model. At this point, a consistency check is carried out. If any errors or inconsistencies are detected in the user's inputs, then error messages are displayed, and the model is not generated. If the program detects something that it thinks is questionable, but is still able to proceed, then it displays warning messages, but proceeds with generating the model.

The REPORT command is used to generate an ASCII file called `report.txt`, describing all the inputs the user has supplied to the program.

The PREPROC command allows the user to graphically inspect the latest model. If the user has changed some parameters after the last GENERATE action, then the PREPROC command detects this and calls the GENERATE command itself. You can select the bodies for which you want to check for results in the SELECT menu in the PREPROC panel. The selected bodies can be visualized in the graphics region by clicking on the DRAWBODIES button.

The SETUP command is used to set up an analysis. The SEPTOL, DSPROF and NPROFDIVS menu can be accessed through the SETUP panel The FEPROBES, SURFGAGES, and LOADSENSORS commands are used to control the data created by the analysis.

The POSTPROC command is used to graphically inspect the results of the analysis.

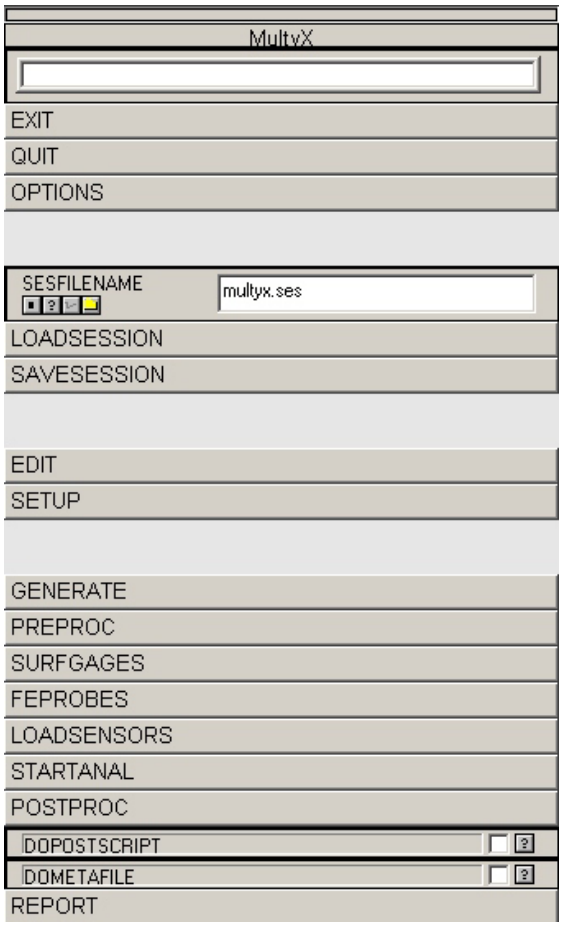


Figure 3.8: The main menu.

Chapter 4

The Graphical User Interface

Planetary2D's user interface is presented by *Guide* in graphical form, as shown in Figure 4.1. *Planetary2D* sends out a stream of informational, error and warning messages to the user. These messages are separated by *Guide*, and presented in separate windows as shown. The user activates these message windows by hitting the appropriate "Error", "Information" or "Warning" tab. Graphical information sent by *Planetary2D* is directed to a graphics window.

4.1 Menu command items

In the example shown in Figure 2.2, the large buttons such as those labeled EXIT, QUIT, OPTIONS, LOADSESSION, EDIT send commands to *Planetary2D* when hit by the user. In response to the command, *Planetary2D* might carry out an action, as in the case of the LOADSESSION command, or lead the user to a different menu, as in the case of the EDIT command. Moving the mouse over a button without depressing it will cause *Guide* to momentarily pop up a balloon (a tool tip) containing a short description of the use of that button. The tool tips can be disabled by the View|DisableToolTips item in the *Guide* main menu.

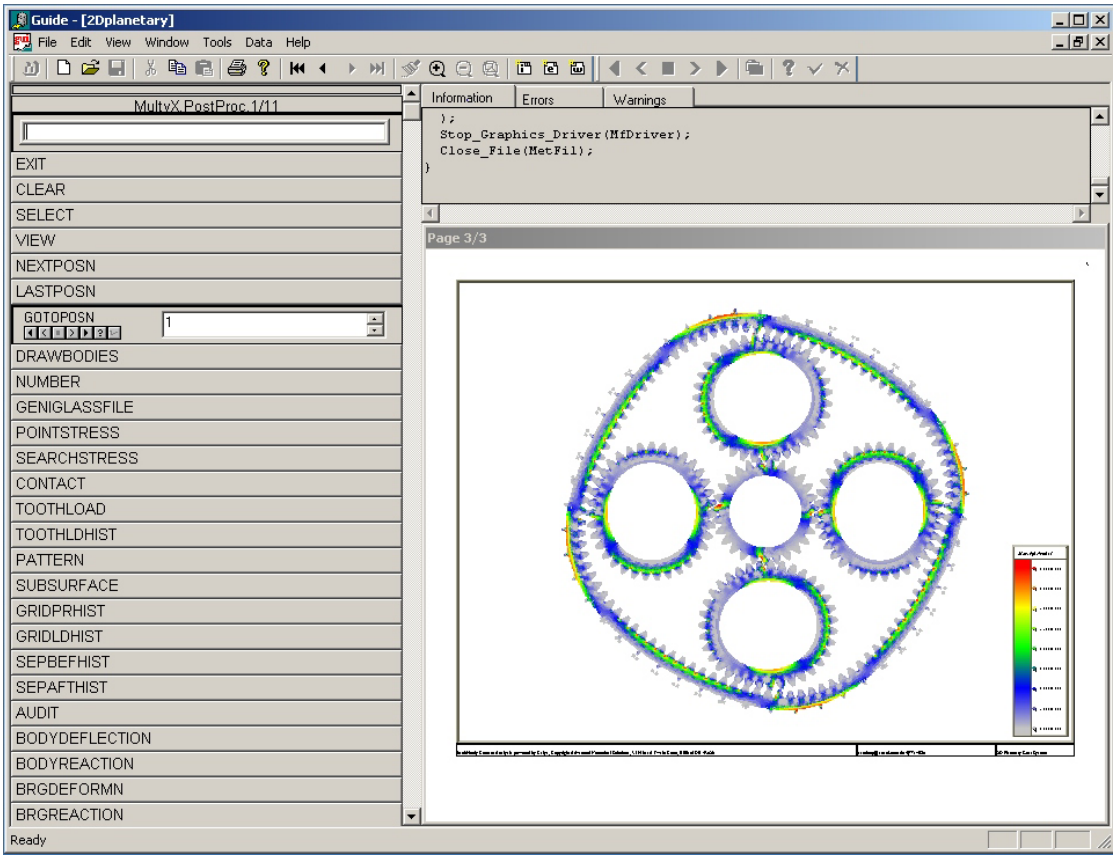


Figure 4.1: Planetary2D user interface

4.2 Integer menu items

Figure 4.2: An integer data entry box

Integer data items are entered through a dialog box of the kind shown in Figure 4.2. The current value appears in a box in the dialog box. If the value of the data item is undefined, then the box appears blank.

4.3 Floating point menu items

Figure 4.3: An floating point data entry box

Floating point data is entered through the dialog box shown in Figure 4.3.

4.4 Boolean menu items

Figure 4.4: An boolean data entry box

Boolean data items are those that can only take a YES/NO or TRUE/FALSE type of value. Their value is set by checking or clearing the box as shown in Figure 4.4.

4.5 String menu items

Figure 4.5: An string data entry box

String data items contain ASCII strings. The dialog box shown in Figure 4.5 allows the user to enter string type data.

4.6 Switch type menu items

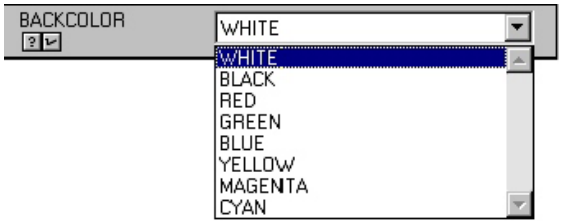


Figure 4.6: An switch type data entry box

The last kind of data item is of the 'switch' type. This item can be switched between a fixed set of valid choices. The choice is made through a drop down list as shown in Figure 4.6.

4.7 Commonly occurring buttons

The data entry dialog boxes use a few small buttons as short cuts for common tasks as shown in the Table 4.1. Some of these buttons may be disabled depending upon the particular item and its value.

Table 4.1: Common buttons

Button	Purpose
	Select the minimum allowable value
	Decrement the value by 1
	Select the default value
	Increment the value by 1
	Select the maximum allowable value
	Accept the value just typed in
	Discard the value just typed in
	Get additional information
	Change the current graphics page
	Change the zoom level
	Refresh the graphics page

4.8 Graphics

Guide directs the graphical output from *Planetary2D* to a graphics window. The graphics are stored as separate pages. A new page is started when *Planetary2D* clears the graphics screen. The user can move between screens using the buttons on the toolbar.

Double clicking anywhere in the graphics window with the left mouse button or dragging the mouse in the graphics window with the left button depressed lets you zoom in. To zoom out, double-click with the right mouse button. The buttons on the toolbar can also be used to zoom in, zoom out and to return to the original view.

By default, the graphics are refreshed automatically when necessary. However, this behaviour can be undesirable if the graphics are very complex. This auto-refresh behavior can be toggled using the View|EnableAutoRefresh and View|DisableAutoRefresh commands. If auto-refresh is disabled, then the user can ask to refresh the graphics using the button.

It is possible to save a sequence of graphics pages in a metafile (a .MET file) using the File|SaveReplayFile command. This file can later be replayed in *Guide* using the File|ReplayGraphicsFile command.

The graphics currently displayed can be saved in Windows Metafile format (a .WMF file) by using the File|SaveWindowsMetafile command. This .WMF file can subsequently be loaded by another application such a word processor. An encapsulated PostScript file (a .EPS file) can be created by using the File|CreateEPSFile command. This command creates an .EPS file containing only the visible part of the current graphics page. Parts of the page that are not visible because of the zoom level will be cropped from the .EPS file.

The Edit|Copy command will copy the graphics in Windows Metafile format onto the clipboard.

Graphics pages can be printed by using the File|Print command on *Guide*'s main menu.

Chapter 5

Building a Model

All the parameters that describe the Planetary gear set are entered in the sub-menus of the EDIT menu (Click EDIT in the main menu to access the EDIT submenus). Figure 5.1 shows the EDIT menu. In the EDIT menu, and in all sub-menus under it, the QUIT command takes the user back to the parent menu after discarding all changes made in the sub-menu and all sub-menus under it. The EXIT command takes the user back to the parent without discarding changes.

There are five submenus under the EDIT menu - SYSTEM, PINION, SUN, RING and CARRIER. The SYSTEM command leads to a menu for entering the system level data. The PINION, SUN, RING and CARRIER commands lead to their respective data specific submenus.

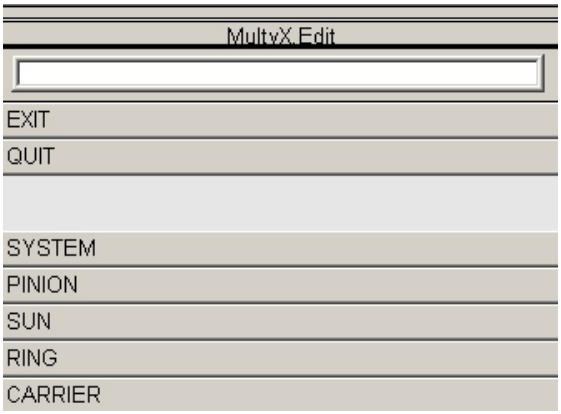


Figure 5.1: The EDIT menu.

5.1 System level data

The SYSTEM command in the EDIT menu of Figure 5.1 leads to the SYSTEM menu shown in Figure 5.2. The parameters in this menu are summarized in Table 5.1.

You can specify the number of pinions in the planetary system using the NUMPINIONS option. The default value is 4. CENTERDIST is the operating center distance. This should always be a positive value.

INPUT switch selects which one of the SUN, RING and CARRIER acts as the input for power. The torque and the angular speed directions of the input member are the same. They are opposite for a member where power is exiting the system. In order to completely specify the kinematics speed of one more member other than the input member is required.

TORQUEINPUT is the torque at the input member (always postive). RPMINPUT is the speed of the input member (sign follows the right hand rule about the z-axis). RPMRING is the speed of the ring(sign follows the right hand rule about the z-axis) and RPMCARRIER is the speed of the carrier(sign follows the right hand rule about the z-axis). If RING is the INPUT, then the system menu asks for RPMINPUT and RPMCARRIER. If the INPUT member is SUN or CARRIER then the system menu asks for RPMINPUT and RPMRING.

The axis of the carrier is assumed to be the origin for the fixed reference frame. Ideally the SUN and the RING gear axes coincide with this origin. However, an assembly error could cause them to be non-coincident. These assembly errors are shown in Figure 5.3

MAGAXISERRSUN and MAGAXISERRRING variables define the magnitude of the sun axis error and ring axis error respectively. ANGAXISERRSUN and ANGAXISERRRING variables define the orientation of the axis error for sun and ring respectively. The value of zero for this angle implies that the error is along the x-axis of the fixed reference frame at time, $t = 0$. MAGRUNOUTSUN and MAGRUNOUTRING variables define the magnitude of the high point of runout error for the sun and ring respectively. ANGRUNOUTSUN and ANGRUNOUTRING variables define the orientation of the high point of runout error for the sun and ring respectively at $t = 0$. A value of zero for this angle implies that the error is along the x-axis of the fixed reference frame at $t = 0$.

If the BACKSIDECONTACT flag is set, then the backside of the teeth will be checked for contact. This flag should be used sparingly because it will cause CPU time to more than double.

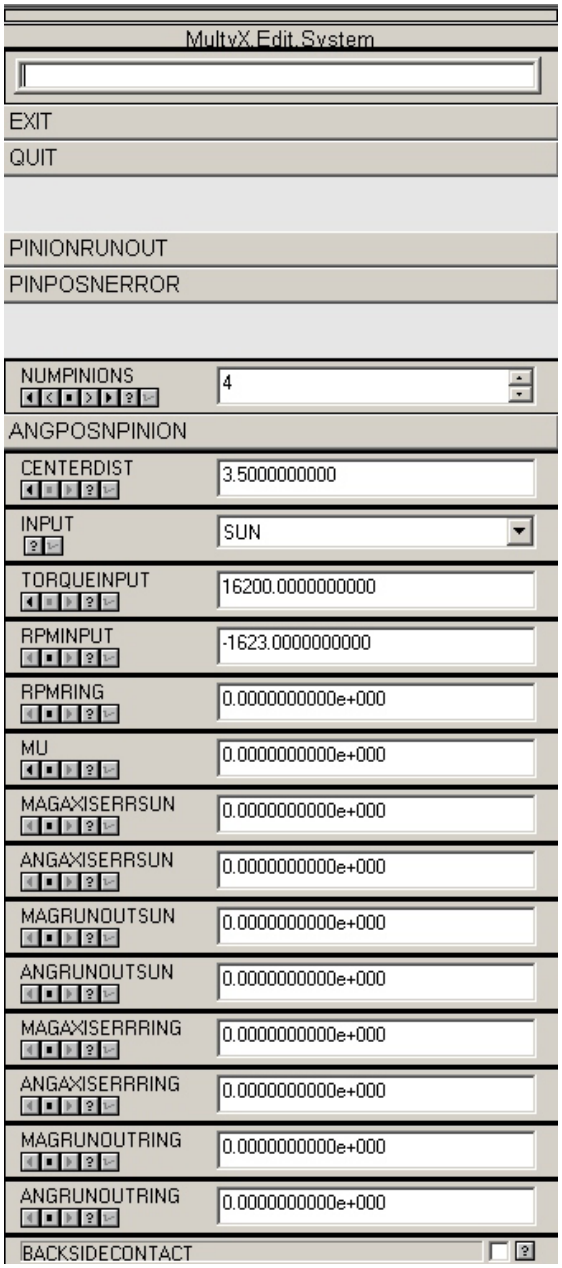


Figure 5.2: The system data menu.

Table 5.1: System configuration parameters

Item	Description
NUMPINIONS	Integer, Number of pinions in the system
CENTERDIST	Float, The operating center distance
INPUT	Switch, Which body is the power input, choices available are SUN, CARRIER and RING
TORQUEINPUT	Float, The torque at the input member
RPMINPUT	Float, Input speed in RPM
RPMRING	Float, The speed of the Ring in RPM (If INPUT $\neq$ RING)
RPMCARRIER	Float, The speed of the Carrier in RPM (If INPUT = RING)
MU	Float, The coefficient of Coulomb friction
MAGAXISERRSUN	Float, Magnitude of Sun axis error
ANGAXISERRSUN	Float, Orientation angle (Deg) of the axis error for Sun
MAGRUNOUTSUN	Float, Magnitude of high point of runout error for Sun
ANGRUNOUTSUN	Float, Orientation angle (Deg) high point of runout error for Sun
MAGAXISERRRING	Float, Magnitude of Ring axis error
ANGAXISERRRING	Float, Orientation angle (Deg) of the axis error for Ring
MAGRUNOUTRING	Float, Magnitude of high point of runout error for Ring
ANGRUNOUTRING	Float, Orientation angle (Deg) high point of runout error for Ring
BACKSIDECONTACT	Boolean, Check for back side contact

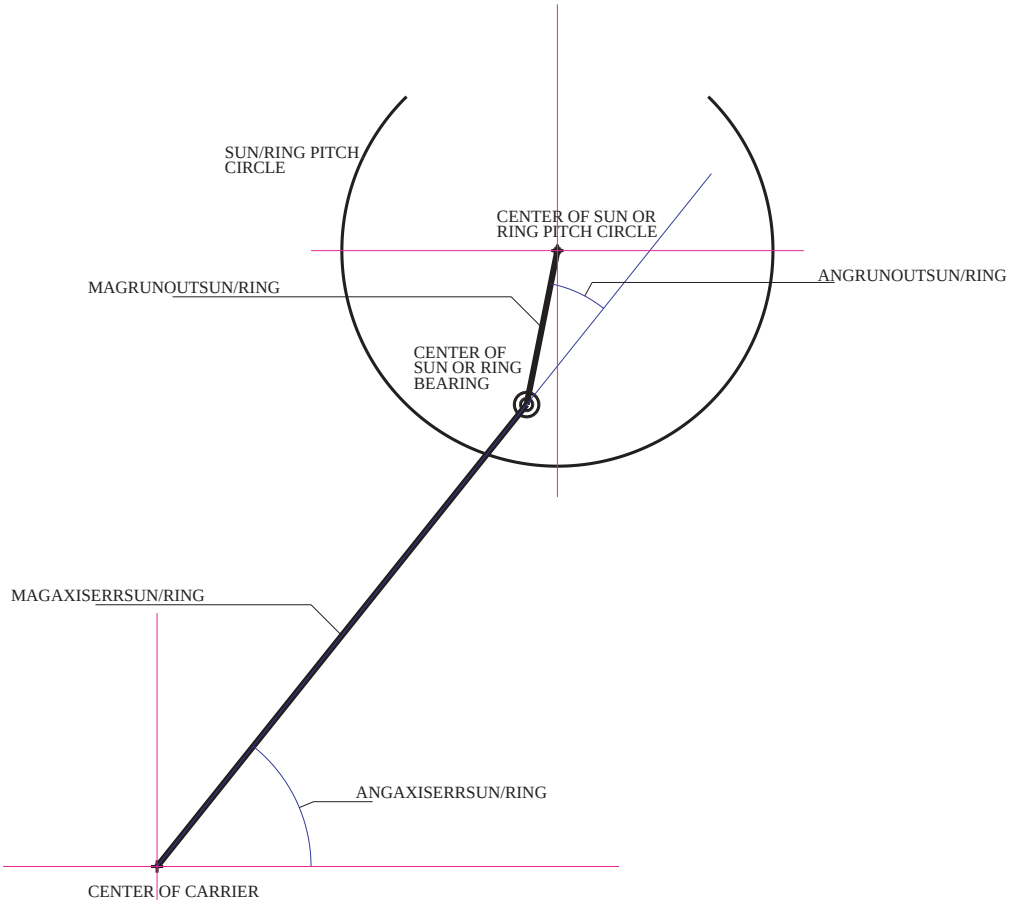


Figure 5.3: Sun and ring gear assembly errors.

Table 5.2: Pinion runout error menu

Item	Description
NUMPINIONS	Integer, Number of pinions in the system
PINIONNO	Integer, Pinion number for which data presently is being displayed
MAGNITUDE	Float, Magnitude of high point of runout error for individual pinions
ANGLE	Float, Orientation Angle (Deg) of the high point of runout error for individual pinions

Table 5.3: Pinion angular position menu

Item	Description
NUMPINIONS	Integer, Number of pinions in the system
PINIONNO	Integer, Pinion number for which data presently is being displayed
ANGPOSNPINION	Float, Nominal Angle (Deg) position for the pinion

5.2 System sub-menus

There are three submenus in the SYSTEM panel- PINIONRUNOUT (Figure 5.5 and Table 5.2), ANGPOSNPINION (Figure 5.6 and Table 5.3) and PINPOSNERROR (Figure 5.7 and Table 5.4).

In the PINIONRUNOUT submenu, you can enter the number of pinions using the NUMPINIONS command. Enter the pinion number for which the data is presently being displayed in the PINIONNO box. For the PINIONRUNOUT panel the MAGNITUDE variable defines the magnitude of the high point of runout errors for individual pinions while the ANGLE variable defines the orientation of the same at time t=0.

In the ANGPOSNPINION panel the ANGPOSNPINION variable defines the nominal angular position of the pinion. Ideally this angular position would be an integer multiple of 360/(NTEETHSUN + NTEETHRING).

The position of the pinion pin or the bearing can be moved by a small amount from the ideal position as shown in Figure 5.4. This pin position error is modelled using the PINPOSNERR command in the SYSTEM menu. In the PINPOSNERR panel the MAGNITUDE variable defines the magnitude of the pin position error for individual pinions while the ANGLE variable defines the orientation of the same for individual pinions. The value of zero for this angle implies that the error is in the radial direction.

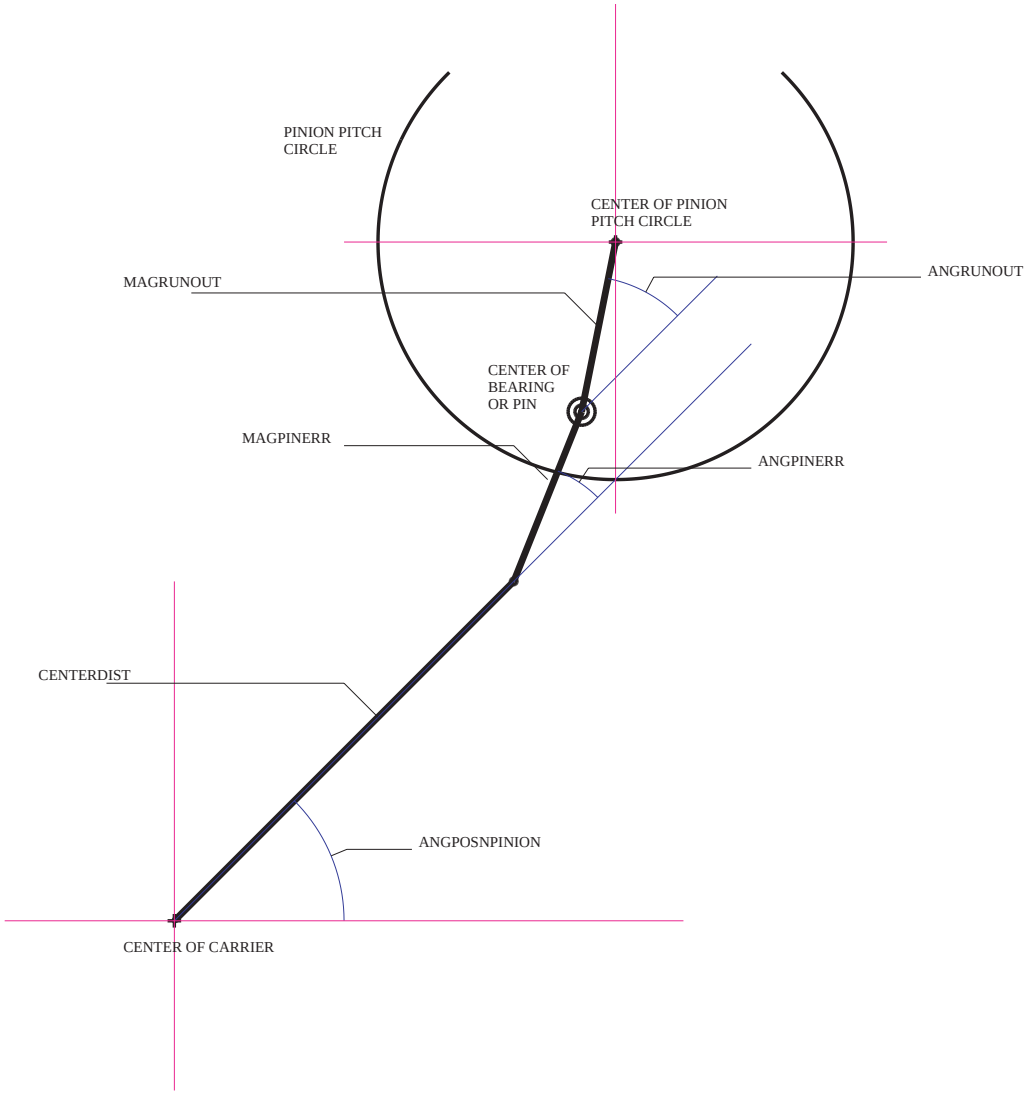


Figure 5.4: The pinion assembly errors.

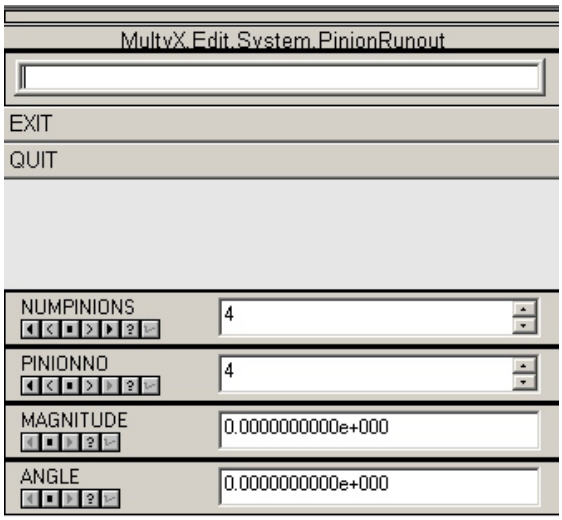


Figure 5.5: The Pinionrunout menu.

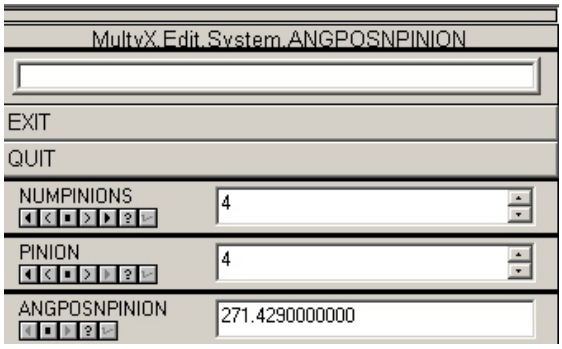


Figure 5.6: The Angposnpinion menu.

Table 5.4: Pin position error menu

Item	Description
NUMPINIONS	Integer, Number of pinions in the system
PINIONNO	Integer, Pinion number for which data presently is being displayed
MAGNITUDE	Float, Magnitude of pin position error for individual pinions
ANGLE	Float, Orientation Angle (Deg) for the pin position error for individual pinions

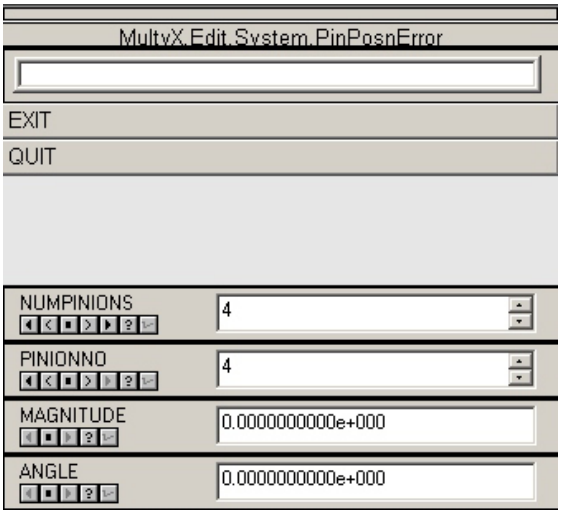


Figure 5.7: The Pinposnerror menu.

5.3 Pinion data

The PINION command in the EDIT menu (Figure 5.1) leads to the Pinion menu shown in Figure 5.8. The three terms - Lumped mass, Lumped moment of inertia and Lump alpha are explained in Table 5.5.

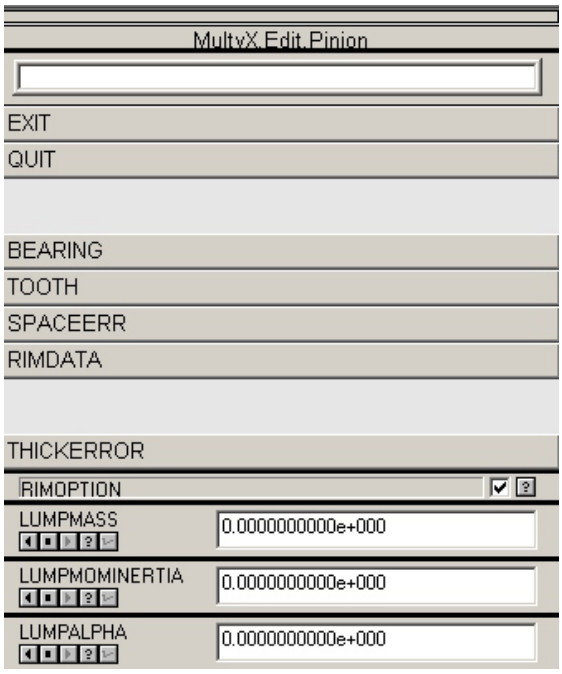


Figure 5.8: The Pinion main menu.

There are five submenus in the Pinion menu - BEARING, SPACEERR, THICKERR, TOOTH and RIMDATA. If the RIMOPTION flag in the PINION menu is set then a separate finite element model will be generated between the RIMDIA of the pinion and the inner diameter.

5.4 Bearings

The BEARING command in Figure 5.8 leads to the Bearing data menu shown in Figure 5.9. Table 5.6 explains the parameters in the Bearing menu. If the rigid race box is checked, then the inner diameter of the pinion is treated as a rigid cylinder. This rigid cylinder will be attached through a 6×6 stiffness matrix and damping matrix to a race on the carrier. Otherwise, its deformation is expressed using a Fourier series expansion in the circular direction. The stiffness and the damping matrices of the bearing are read in from an existing input data file. The name of the file should be entered in to the BRGFILE box.

The contents of a sample bearing file are shown below.

```
0 0 0 0 0 0
1.0e7 0 0 0 0 0
0 1.0e7 0 0 0 0
0 0 0 0 0 0
0 0 0 0.0e9 0 0
0 0 0 0 0.0e9 0
```

Table 5.5: Pinion lumped parameters

Item	Description
LUMPMASS	Float, Lumped mass added at the center of the pinion in addition to the mass of the tooth and rim model
LUMPMOMINERTIA	Float, Lumped polar moment of inertia Jz added at the center of the pinion in addition to the Jz of the tooth and rim model
LUMPALPHA	Float, The damping constant for the lumped mass and moment of inertia

Table 5.6: Bearing menu parameters

Item	Description
RIGIDRACE	Boolean, Whether the bearing race is a rigid cylinder
CIRCORDER	Integer, Fourier series order in the circular direction
BRGFILE	String, Bearing file name for pinion bearing

```
0 0 0 0 0 0
0 0 0 0 0 0
0 0 0 0 0 0
0 0 0 0 0 0
0 0 0 0 0 0
0 0 0 0 0 0
0 0 0 0 0 0
0
```

Inside the bearing file, the first line in this file must always have 6 zeroes. This line is assigned for future use. The next 12 lines contain the $6x6$ stiffness and 6×6 damping matrices. The last line contains a zero. This is a flag intended for future use. The $6x6$ matrices correspond to the 6 degrees of freedom of bearing race 1 relative to bearing race 2, as measured in the race 2 reference frame.

The six degrees of Freedom are the three translations in the X, Y and Z directions and the three rotations about the X, Y and Z directions. For the Pinion bearing the X-direction is always in the radial direction pointing from the carrier axis to the pinion axis. The Y-axis points in the tangential direction.

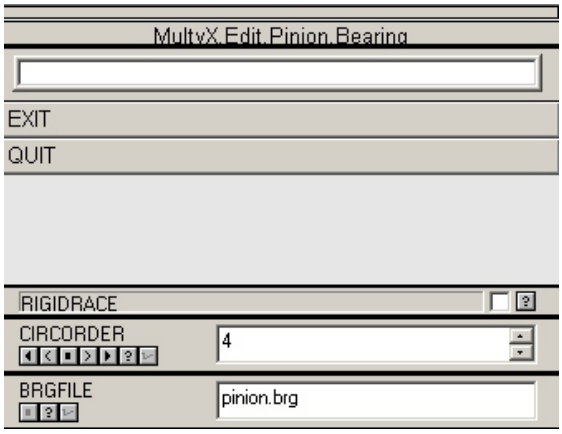


Figure 5.9: The Bearing menu.

5.5 Tooth numbering

Each individual tooth of the Pinion, Sun and Ring gear is assigned a tooth number. Figure 5.10 shows the numbering scheme used for the Pinion and Sun gear. Figure 5.11 shows the numbering scheme used for the Ring gear.

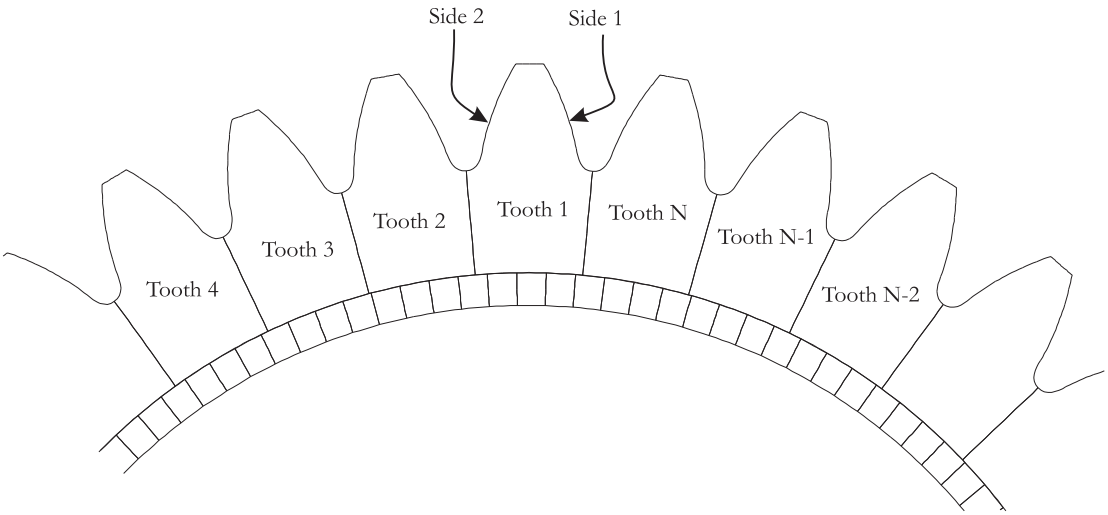


Figure 5.10: The tooth and side numbering scheme for the pinion and sun gear

The two sides of each tooth are labelled, SIDE1 and SIDE2 as shown in Figures 5.10 and 5.11.

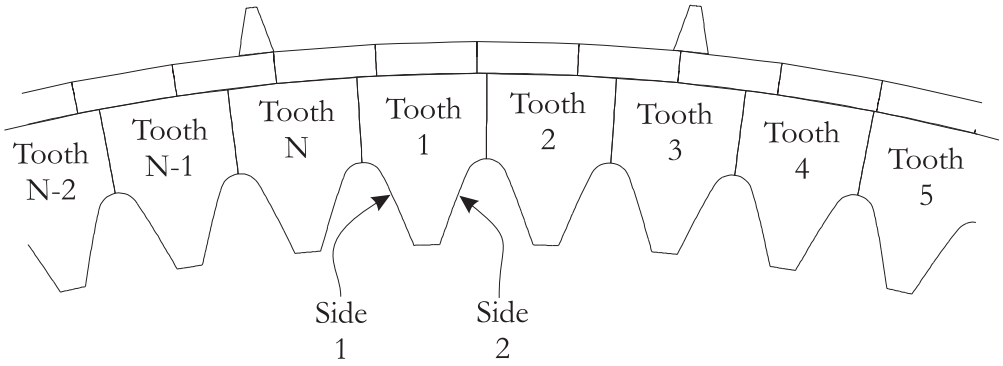


Figure 5.11: The tooth and side numbering scheme for the ring gear

5.6 Spacing error menu

The SPACEERR command in Figure 5.8 leads to the Spacing error menu shown in Figure 5.12. The spacing error is an angular amount by which the two surfaces of individual tooth on the individual pinion can be rotated from their nominal positions. This value is in radians and corresponds to a positive rotation of the tooth surface about the pinion z-axis, following the right hand rule.

SIDE1 and SIDE2 are the two sides of each tooth. If you click any of these it will lead you to a separate submenu as shown in Figure 5.13. Table 5.7 explains the parameters in the Spacing error menu.

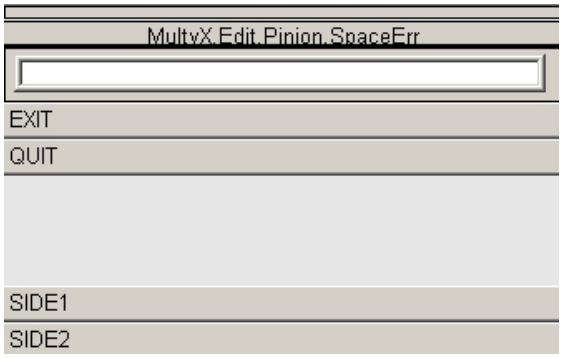


Figure 5.12: The Spacing error menu.

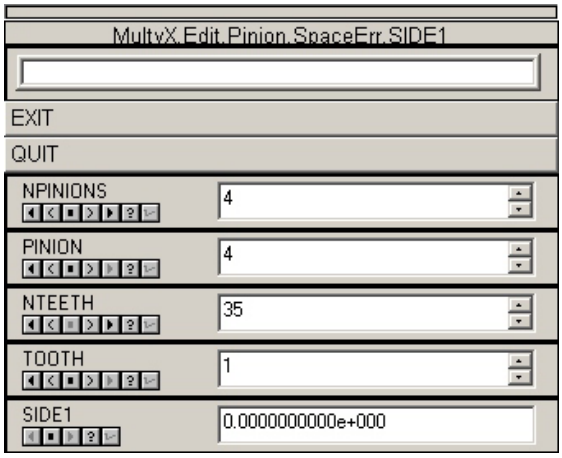


Figure 5.13: The Spacing error submenu.

Table 5.7: Spacing error menu

Item	Description
NPINIONS	Integer, Number of pinions in the system
PINION	Integer, Pinion number for which data presently is being displayed
NTEETH	Integer, Number of teeth on the pinion
TOOTH	Integer, The tooth number on the pinion
SIDE1	Float, Spacing error on SIDE1 of individual teeth of the pinion

5.7 Thickness errors

The THICKERROR command in Figure 5.8 leads to the thickness error menu shown in Figure 5.14. The thickness error is the amount by which the circular tooth thickness at the pitch circle of a particular pinion is more than the nominal amount. A different thickness error can be prescribed for each pinion. The thickness error is applied to the tooth by rotating the contact surfaces about the pitch circle center.

Table 5.8 explains all the thickness error parameters.

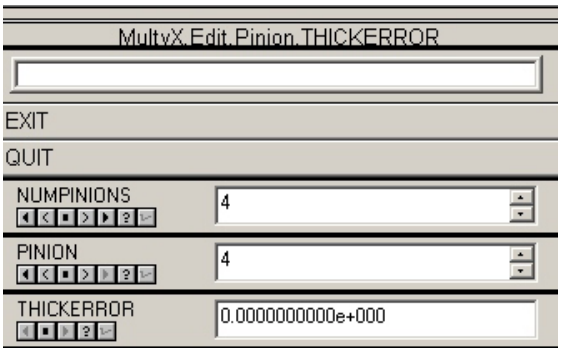


Figure 5.14: The Thickness error menu.

Table 5.8: Thickness error menu

Item	Description
NUMPINIONS	Integer, Number of pinions in the system
PINIONNO	Integer, Pinion number for which data presently is being displayed
THICKERROR	Float, Thickness error in individual pinions

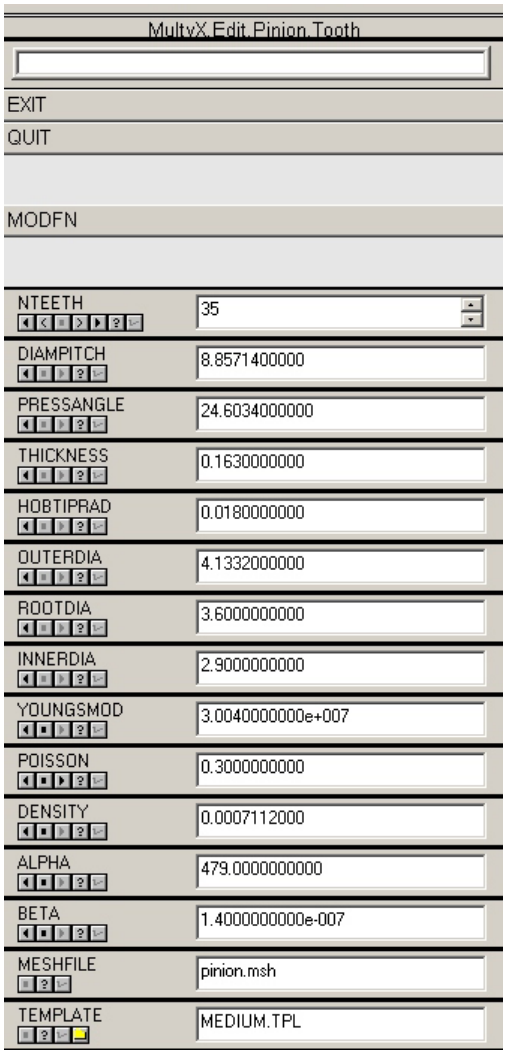


Figure 5.15: The tooth data menu for the pinion gear.

5.8 Modelling the tooth profile

The TOOTH command in Figure 5.8 leads to the tooth data menu shown in Figure 5.15. Table 5.9 describes the parameters required for modelling the pinion tooth. In this table C,M and K are the damping, mass and stiffness matrices.

Figure 5.16 shows the details of the various diameters definitions for an external gear.

The Mesh file contains element connectivity and geometry information. This file is created by the program. The complete path of the file should be entered if the file is not to be created in the current directory. The file extension should also be provided.

The Template file contains element connectivity information. The complete path of the file should be entered if the file is not in the current directory. The file extension should also be provided. There are several pre-built template files. The element numbering used in the pre-built template files is described in Appendix A. Medium.tpl (Figure A.1) is the most commonly used template file for mesh generation. Very good stress predictions in the root region are obtained

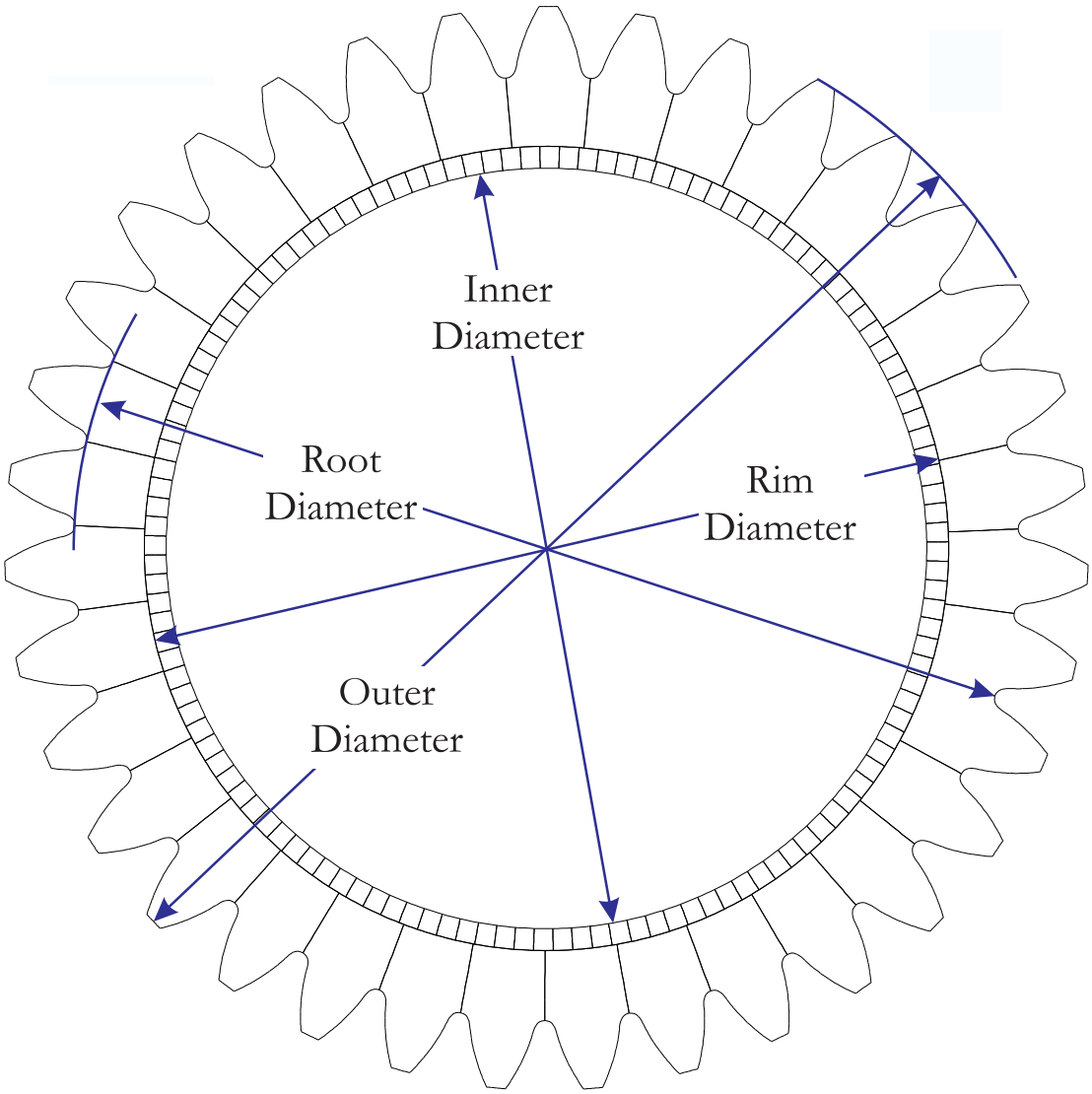


Figure 5.16: Diameter definitions for an external gear.

Table 5.9: Parameters describing the tooth profile for pinion

Item	Description
NTEETH	Integer, The number of teeth on pinion
DIAMPITCH	Float, Diametral pitch of the pinion
PRESSANGLE	Float, Pressure angle (deg) of the pinion
THICKNESS	Float, Transverse thickness of the pinion tooth
HOBTIPRAD	Float, Tip radius of the hob used to generate the pinion
OUTERDIA	Float, Outer diameter of the pinion tooth
ROOTDIA	Float, Root diameter of the pinion tooth
INNERDIA	Float, Inner diameter of the pinion rim
YOUNGSMOD	Float, Young's modulus for the pinion
POISSON	Float, Poisson's ratio of the pinion
DENSITY	Float, Density of the pinion
ALPHA	Float, Value of alpha in the equation - C = alpha * M + beta * K
BETA	Float, Value of beta in the equation - C = alpha * M + beta * K
MESHFILE	String, Mesh file name for pinion
TEMPLATE	String, Template file name for pinion

using this file since higher order elements are used in that region. For convergence studies use the finer template files (Figure A.2 and A.3). The element shapes generated by these template files are optimal when the difference (OUTERDIA - ROOTDIA) is approximately same as the difference (ROOTDIA - RIMDIA). If this is not possible (when RIMDIA is too large), a special template file called THINRIM.tpl (Figure A.4) has to be used. This template generates fewer elements in order to maintain an adequate aspect ratio.

Table 5.10: Rayleigh damping parameters for steel.

Material	α (1/sec)	β (sec)
Steel	479	1.2e-07

5.9 Rayleigh Damping Model

The Rayleigh Damping model assumes that the damping matirx C for a finite element is is related to the mass matrix M and the stiffness matrix K by a linear relationship.

$$C = \alpha M + \beta K \tag{5.1}$$

α has units of `1/sec` and β has units of `sec`. The relationship between the Rayleigh damping model and the equivalent viscous damping model is obtained by from a single degree of free system, by comparing the two equivalent damping models.

$$m\ddot{x} + c\dot{x} + kx = f(t) \tag{5.2}$$

Comparing the two damping models yields,

$$\begin{aligned} \frac{c}{m} &= 2\zeta\omega_n = \alpha + \beta\omega_n^2 \\ 2\zeta &= \frac{\alpha}{\omega_n} + \beta\omega_n \end{aligned} \tag{5.3}$$

In a general multi-degree of freedom system, if the damping ratios ζ_i are measured at a number of system natural frequencies ω_i, the paramters for the Rayleigh damping model can be computed using a least squares approach.

$$\begin{Bmatrix} 2\zeta_1 \\ 2\zeta_2 \\ \vdots \\ 2\zeta_i \\ \vdots \\ 2\zeta_n \end{Bmatrix} = \begin{bmatrix} \frac{1}{\omega_1} & \omega_1 \\ \frac{1}{\omega_2} & \omega_2 \\ \vdots & \vdots \\ \frac{1}{\omega_i} & \omega_i \\ \vdots & \vdots \\ \frac{1}{\omega_n} & \omega_n \end{bmatrix} \begin{Bmatrix} \alpha \\ \beta \end{Bmatrix} \tag{5.4}$$

$$2\{\zeta\} = [A] \begin{Bmatrix} \alpha \\ \beta \end{Bmatrix} \tag{5.5}$$

Equation 5.5 is solved to compute α and β. Least squares solution is used if the values of damping ratio in Equation 5.5 are measured at more than two different natural frequencies. In general, Equation 5.5 is solved in the regular sense, by measuring the damping ratio at two natural frequencies of the system. Table 5.10 shows α and β for steel.

5.9.1 Modal Damping Ratio - SDOF system

The displacement frequency response function for a single degree of freedom system is as follows.

$$\left|\frac{x}{x_{static}}\right| = \frac{1}{(1-r^2)^2 + 4\zeta^2 r^2} \tag{5.6}$$

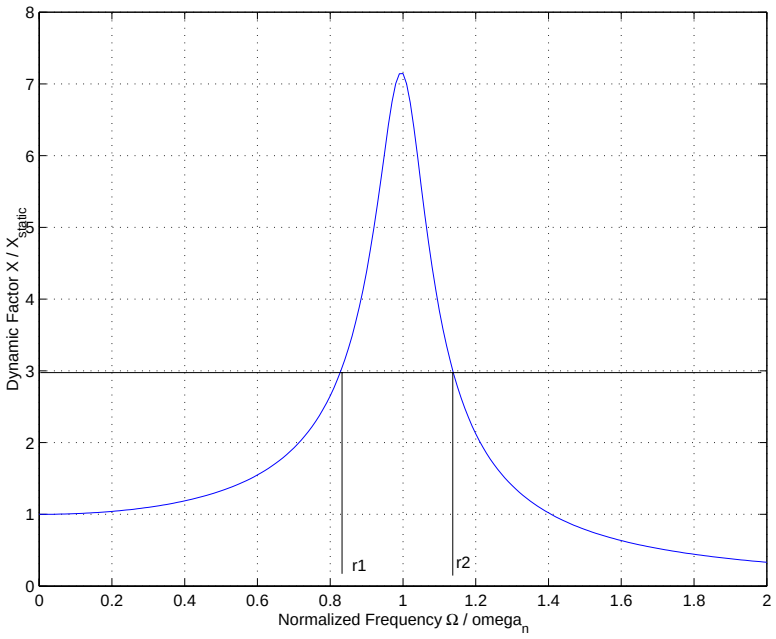


Figure 5.17: Frequency Response of a SDOF system.

where $r = \Omega/\omega_n$ is ratio of the excitation frequency to the natural frequency, and $x_{static} = F/K$ is the static deflection. Figure 5.17 shows the frequency response of a sdof system. At resonance, the response is controlled purely by damping $|\frac{x_p}{x_{static}}| = \frac{1}{2\zeta}$. The width of the peak depends on the damping ratio. The damping ratio can be estimated as follows - Let $r_1 = \Omega_1/\omega_n$ and $r_2 = \Omega_2/\omega_n$ be the frequencies where the dynamic deflection level is less than the peak value, by a factor of n, that is $|\frac{x_r}{x_p}| = \frac{1}{n}$. Two estimates of the damping ratio is obtained as

$$\zeta^{(1)} = \frac{1 - r_1^2}{2\sqrt{n^2 - r_1^2}} \tag{5.7}$$

$$\zeta^{(2)} = \frac{r_2^2 - 1}{2\sqrt{n^2 - r_2^2}} \tag{5.8}$$

The system damping ratio is ζ is the average value of the above two estimates.

$$\zeta = \frac{\zeta^{(1)} + \zeta^{(2)}}{2} \tag{5.9}$$

5.9.2 Modal Damping Ratio Measurement

The modal damping ratio (based on viscous damping model) at various system natural frequencies is computed from the measured frequency response function (**driving point compliance**). The dynamic displacement response at dof k due to force at dof j is given as

$$\frac{x_k}{f_j}(\Omega) = \sum_{i=1}^{n} \frac{U_{ik}U_{ij}}{-\Omega^2 + 2j\zeta_i\omega_i\Omega + \omega_i^2} \tag{5.10}$$

where the summation index i goes from mode 1 to mode n. The modal vectors U_i are normalized with respect to the mass matrix. U_{ik} is the k-th element of the the i-th modal vector. The driving

point compliance ($k = j$) is guarenteed to have an antiresonance between two resonances, since the numerator terms are always positive and the denominator terms are of opposing signs when the excitation frequency is between two resonances.

The driving point compliance of a gear is measured from an impulse test. The modal damping ratio of each mode is computed based on the sharpness of the peak at the natural frequency, using Equations 5.7, 5.8 and 5.9. Once the modal damping ratio for a number of system modes are computed, Equation 5.5 is used to determine the parameters for the Rayleigh damping model.

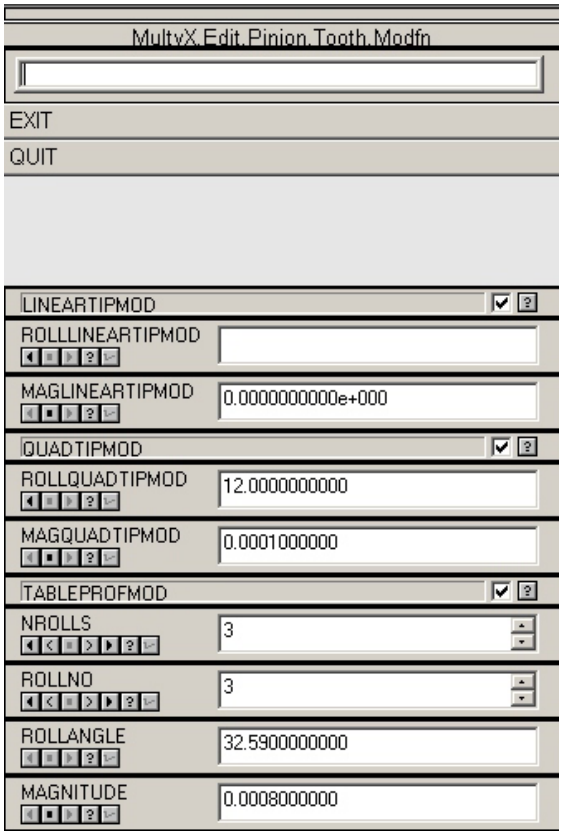


Figure 5.18: The tooth modification menu.

5.10 Surface modifications

The MODFN command in the tooth data menu (Figure 5.15) leads to the menu shown in Figure 5.18. This menu is used to specify surface modifications. Three simple surface modifcations can be specified directly in this menu, linear tip modification, quadratic tip relief, and tabular surface modification.

Linear tip modification is applied using the parameters shown in Table 5.11 and Figure 5.19. The modification magnitude is a linear function of the involute roll angle. Positive value for the magnitude of linear tip modfn. indicates removal of material.

Quadratic tip modification is applied using the parameters shown in Table 5.12 and Figure 5.20. The modification magnitude is a quadratic function of the involute roll angle. A positive value for the magnitude implies removal of material.

Tabular surface modification is applied using the parameters shown in Table 5.13 and Figure 5.21. A table of roll angle and modification values will be used to describe the profile modification. The profile modification is linearly interpolated between the roll angle values. A positive value for magnitude indicates removal of material.

Table 5.11: Linear tip modification parameters

Item	Description
LINEARTIPMOD	Boolean, Whether to apply the linear tip modfn.
ROLLLINEARTIPMOD	Float, Roll angle (Deg) at start of the linear tip modfn.
MAGLINEARTIPMOD	Float Magnitude of the linear tip modfn.

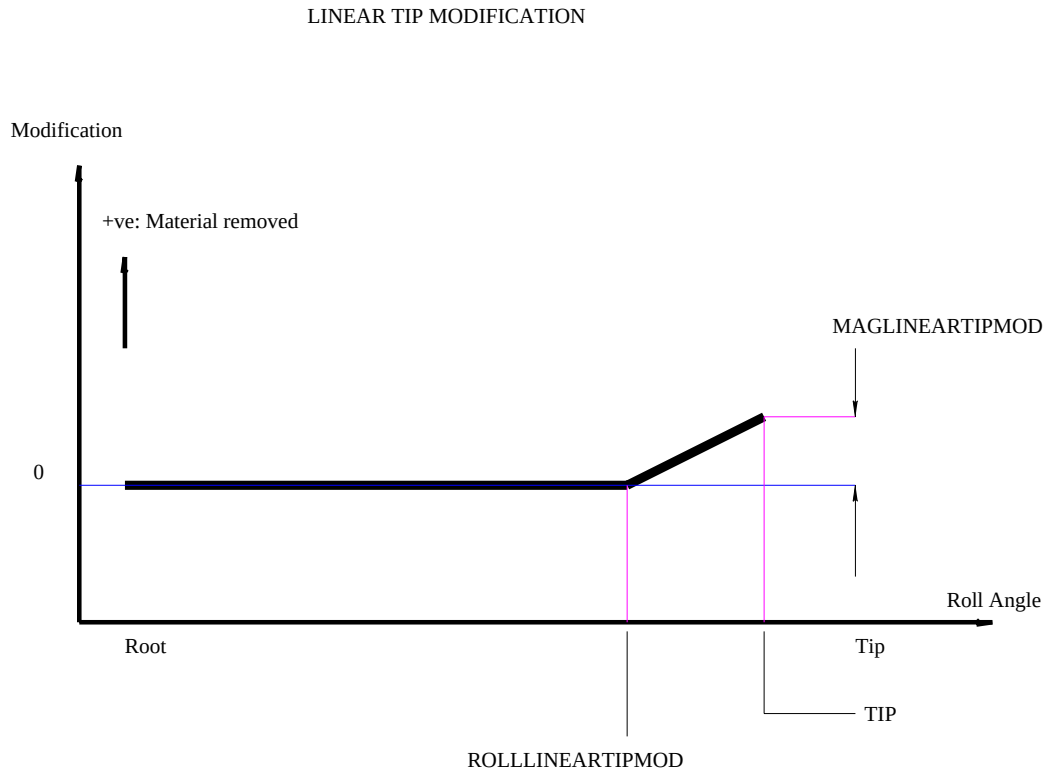


Figure 5.19: Linear tip modification

Table 5.12: Quadratic tip modification parameters

Item	Description
QUADTIPMOD	Boolean Whether to apply the quadratic tip modfn.
ROLLQUADTIPMOD	Float Roll angle (Deg) at start of the quadratic tip modfn.
MAGQUADTIPMOD	Float Magnitude of the quadratic tip modfn.

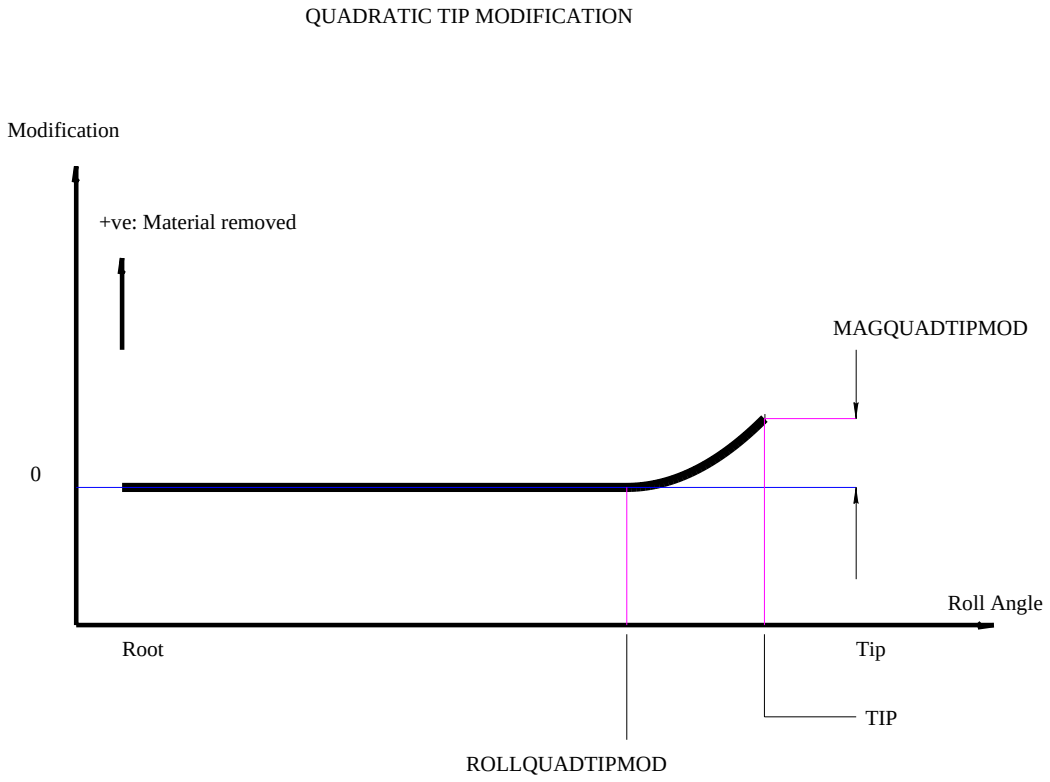


Figure 5.20: Quadratic tip modification

Table 5.13: Tabular profile modification parameters

Item	Description
TABLEPROFMOD	Boolean, Whether to use a profile modfn table.
NROLLS	Integer, The number roll angles used in the profile modfn. table
ROLLNO	Integer, The roll angle number in the profile modfn. table
ROLLANGLE	Float, Vector, indexed by IROLL, Roll angle in the profile modfn. table
MAGNITUDE	Float, Vector, indexed by IROLL, Magnitude of modification in the profile modfn. table

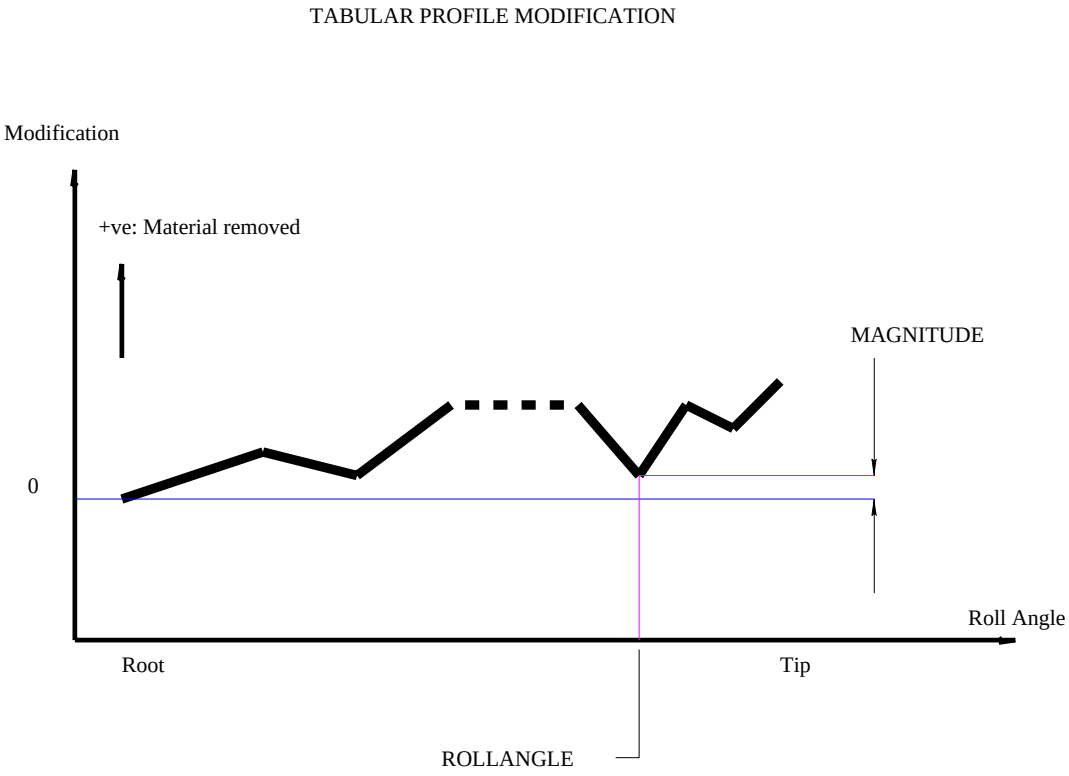


Figure 5.21: Tabular tip modification

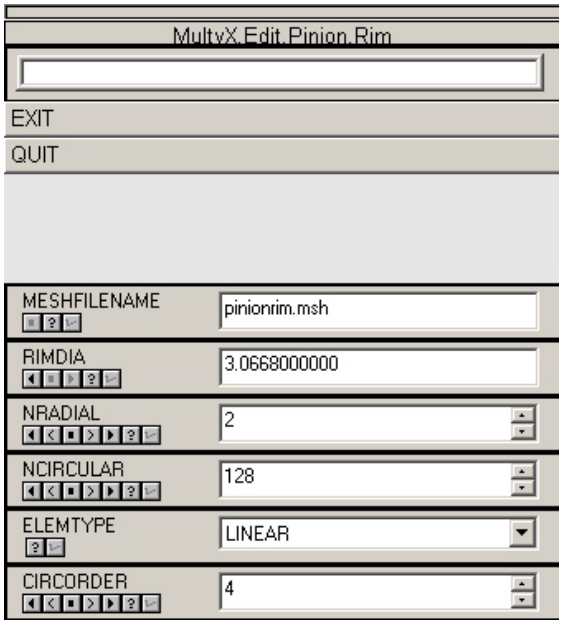


Figure 5.22: The menu for specifying rim data for pinion.

5.11 Modelling the rim

The RIMDATA command in the pinion data menu (Figure 5.8) leads to the rimdata menu shown in Figure 5.22. Table 5.14 explains the rim data parameters for the pinion.

There are three types of element types you can use to generate the pinion rim mesh-Linear, Quadratic and Cubic. The mesh file contains the element connectivity and geometry information. This file is created by the program. The complete path of the file should be entered if the file is not to be created in the current directory. The file extension should also be provided. Changing the rim diameter affects the aspect ratio of the elements. The standard template files are designed to create the best aspect ratio when RIMDIA = 2 * ROOTDIA - OUTERDIA. The deformation of the rim at the interface between the tooth and rim model is expressed as a Fourier series. If the pinion bearing race is modeled as a flexible cylinder, then the bearing race is modeled using the Fourier shape functions in the circular direction. The CIRCORDER variable specifies the order of the Fourier series in the circular direction.

Table 5.14: Rim data parameters

Item	Description
MESHFILENAME	String, Mesh file name for pinion rim
RIMDIA	Float, Rim diameter of the pinion rim
NRADIAL	Integer, Number of finite elements in the radial direction in the rim
NCIRCULAR	Integer, Number of finite elements in the circumferential direction in the rim
ELEMTYPE	Switch, Element type to be used for pinion rim
CIRCORDER	Integer, Fourier series order in the circular direction

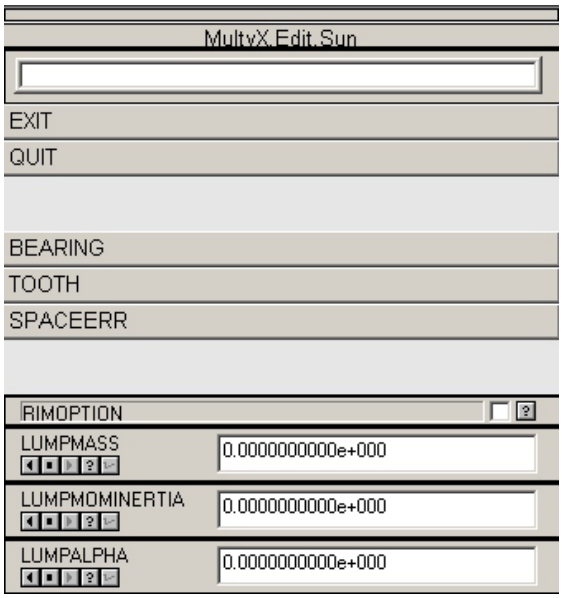


Figure 5.23: The Sun gear menu.

5.12 Sun gear data

The SUN command in the EDIT menu (Figure 5.1) leads us to the sun data menu as shown in Figure 5.23. The SUN menu is similar to the PINION menu except that there are no Thickness error inputs. If the RIMOPTION flag is set then a separate finite element model will be generated between the Rim Dia of the Sun gear and the inner dia.

5.13 Sun gear bearings

The Bearing menu for the Sun gear is shown in Figure 5.24. The menu is similar to the Pinion bearing menu except for the BEARING command which is not there in the pinion bearing menu. If this BEARING flag is set, then the sun has a bearing and a bearing file has to be provided. Otherwise the Sun can be made to float or can be constrained. The CONSTRAINT switch (if the Bearing box is not checked) decides whether the sun is allowed to float or is held at its axis.

The SPACEERR, TOOTH and the RIMDATA menus for the sun gear are similar to those for the pinion.

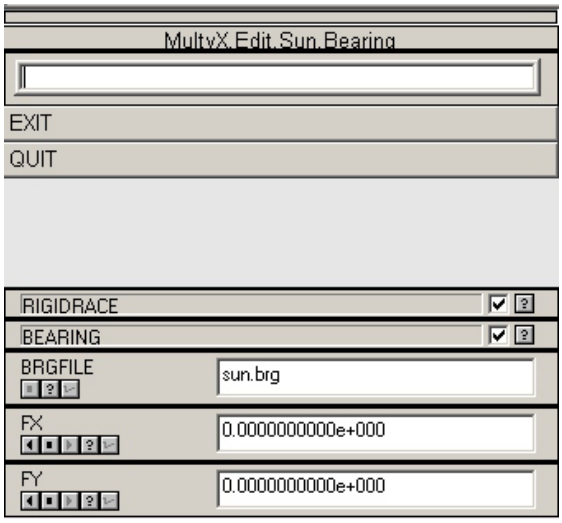


Figure 5.24: The Sun bearing menu.

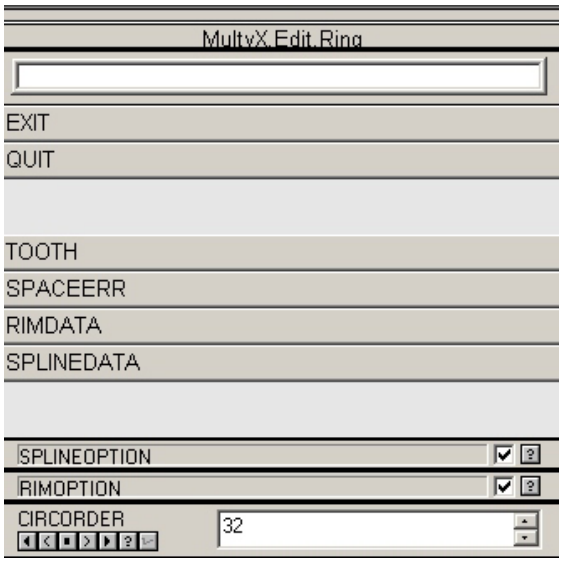


Figure 5.25: The Ring data menu.

5.14 Ring data

The RING command in the EDIT menu (Figure 5.1) leads to the ring data menu shown in Figure 5.25.

If the SPLINEOPTION is checked, SPLINEDATA comand will show up in the ring menu (Figure 5.25). Ring gear splines will be modeled if you check this option. These splines are on the outer diameter on the ring gear. Click on the SPLINEDATA button to access its menu (Figure 5.26). Table 5.15 explains the parameters in the SPLINEDATA menu. If you do not select the SPLINEOPTION then a menu called RIGIDOUTERDIA pops up. If you check this option then the rim outer diameter is modeled as a rigid body.

If the RIMOPTION is selected then a separate finite element model will be generated between the rim diameter of the ring gear and the outer diameter. RIMDATA pops up if the RIMOPTION box is checked. Click on the RIMDATA command (Figure 5.25) to access the rim data menu. The ring rim parameters are similar to the pinion rim parameters.

The TOOTH menu and the SPACEERR menu for the Ring data are similar to those in the PINION menu.

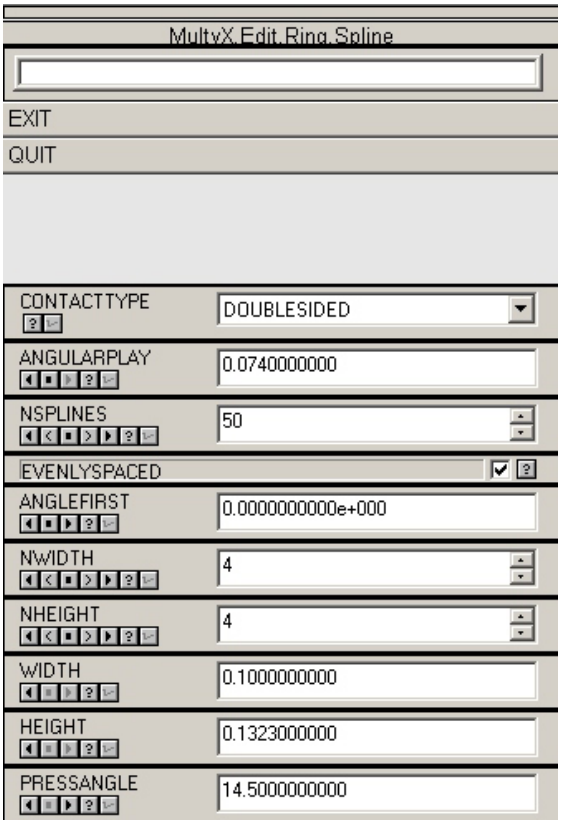


Figure 5.26: The Spline data menu.

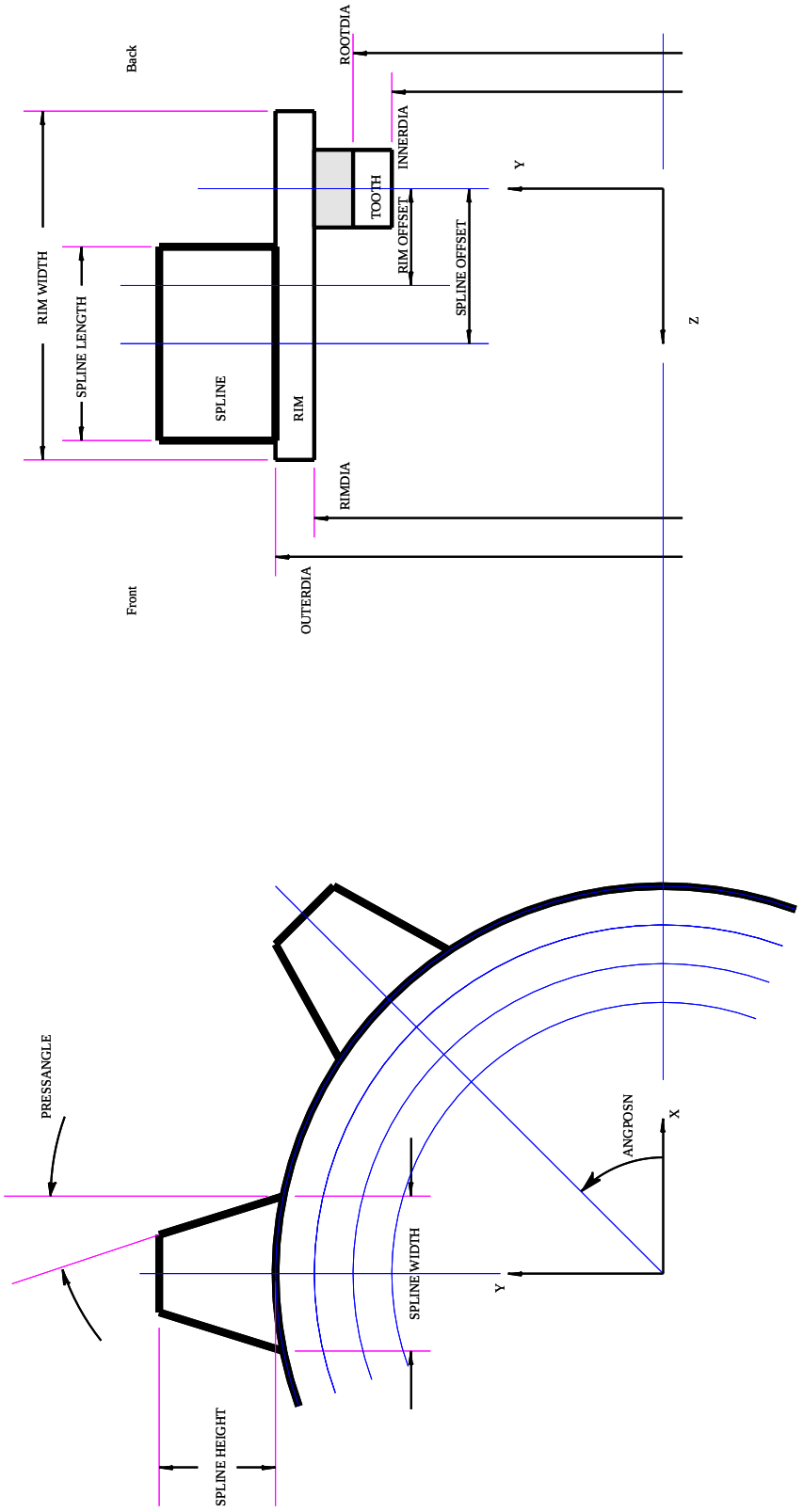


Figure 5.27: Ring gear spline geometry

Table 5.15: Spline data parameters

Item	Description
CONTACTTYPE	Switch, Whether contact conditions are enforced on both sides of the spline or just one side
ANGULARPLAY	Float, Amount of play (deg) allowed to the ring gear due to clearance at the splines
NSPLINES	Integer, Number of splines in the rim gear diameter
EVENLYSPACED	Boolean, Whether splines are evenly spaced
ANGLEFIRST	Float, Angular position of the first spline
NWIDTH	Integer, Number of elements NWIDTH across the span as shown in Figure 5.27
NHEIGHT	Integer, Number of elements NHEIGHT across the span as shown in Figure 5.27
WIDTH	Float, Spline width
HEIGHT	Float, Spline height
PRESSANGLE	Float, Spline pressure angle

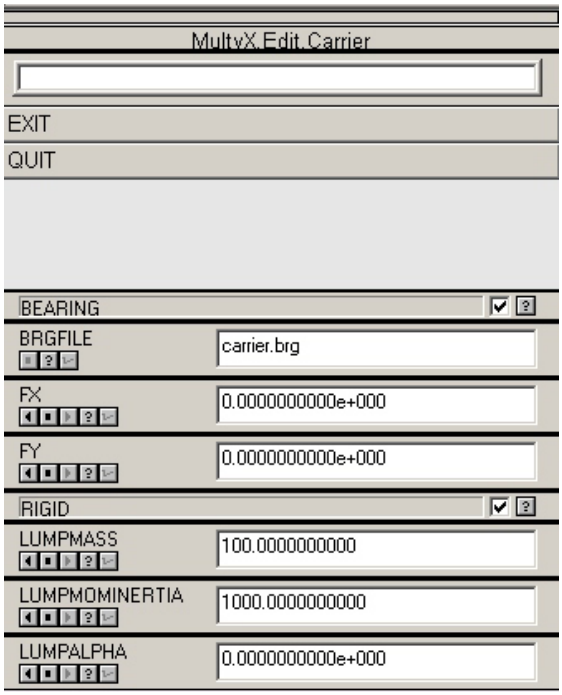


Figure 5.28: The carrier menu.

5.15 Carrier data

The CARRIER command in the EDIT menu (Figure 5.1) leads to the CARRIER menu shown in Figure 5.28. Table 5.16 explains the terms associated with the CARRIER menu. If you set the RIGID box in the Carrier menu then the Carrier is modeled as a rigid body, otherwise it will be modeled using a stiffness matrix. If you set the BEARING flag in the Carrier menu then the Carrier has a bearing and a bearing file name has to be provided. Otherwise the Carrier can be made to float or can be constrained.

Table 5.16: Carrier data parameters

Item	Description
BEARING	Boolean, Whether the carrier has a bearing
BRGFILE	String, Bearing file name if carrier has a bearing
CONSTRAINT	Switch, Whether the carrier is floating or is fixed
RIGID	Boolean, Whether the carrier is modeled as a rigid body
STFFILE	String, Stiffness file name for the carrier
LUMPMASS	Float, Additional lump mass at carrier center
LUMPMOMINERTIA	Float, Lumped polar moment of inertia Jz at the carrier center
LUMPALPHA	Float, Damping constant alpha for the lumped mass

Chapter 6

Running an Analysis

The analysis is started by using STARTANAL command of Figure 3.8.

Before starting an analysis, sensor locations have to be set up to measure stress and loads in the model. This is done through the SURFGAGES, FEPROBES and LOADSENSORS commands in the main menu (Figure 3.8). Additional analysis parameters and settings are controlled through the SETUP command.

6.1 Surface gages

A surface gage is used to measure the critical stress along tooth surfaces. The reading of each gage is the most critical stress measured over a user defined range of teeth, profile, face and depth along a specific surface.

Figure 6.1 shows the Surface Gage setup menu. The number of gages NGAGES has to be entered first. Then the gage number for a particular gage can be entered into the GAGE box, and the gage information can be typed into the remaining boxes. For each gage, the BODY item selects which of the individual components in the system the gage is attached to. There is a switch provided to select the individual component. For a planetary gear set, the bodies are the Sun, the Ring, the individual Pinions, the Housing(if splines are used) and the Carrier.

After the Body is selected, the surface on which the gage should be attached should be selected. The sun and pinion gear teeth typically have four surfaces. SURFACE1 and SURFACE2 cover the entire involute and fillet areas of the two sides Side 1 and Side 2, respectively, of the teeth. FILLET1 and FILLET2 cover only the fillet region of Side 1 and Side 2, respectively. If the ring gear is selected as the body then there are two more surfaces you can select, SPLINESURF1 and SPLINESURF2 which are the two sides of the spline teeth on the ring gear diameter. If the housing is selected as the body then there are only two surfaces, SPLINESURF1 and SPLINESURF2.

When there are multiple copies of a surface on a body, each individual copy of that surface is called an instance of that surface, and is given a unique instance number. In the case of gear tooth surfaces, the instance number is the same as the tooth number. The parameters TOOTHBEGIN and TOOTHEND define a range of teeth over which the gages will be placed. The reading of the gage is the stress at the most critical tooth. If the value of TOOTHBEGIN is greater than TOOTHEND, then the search range will wrap around the last tooth.

There are two parameters that identify a point on a surface. We refer to these two parameters as S which varies in the profile direction, and T which varies in the face width direction. In a 2D model the parameter T is ignored.

The profile parameter S increases from fillet to the tip on Side 1 of a tooth, and from the tip to the fillet on Side 2, as shown in Appendix A. The parameters SPROFBEGIN and SPROFEND

define a range over which the stress will be calculated. These are in surface local units as shown in Appendix A. The GAGE will read out the critical value of stress in this range. The NUMSPROF parameter controls how many search points should be used over this range.

The face width range parameters TFACEBEGIN, TFACEEND and NUMTFACE are ignored for 2D models.

The DEPTHBEGIN, DEPTHEND and NUMDEPTH parameters extend the search range to a number NUMDEPTH of points ranging in depth from DEPTHBEGIN to DEPTHEND below the surface. This is an expensive computation, and should not be used unless necessary. The surface gage will measure the stress at the critical depth. The depth is in physical length units.

Because finite element stresses computed very close to the highly concentrated contact loads can have a large amount of error, we need a way to screen out points that are too close. The parameter DISTMIN is the minimum allowed distance of a stress calculation point from a contact point. Stresses will not be calculated at any point whose distance from a contact point is less than this value. This distance is in physical length units.

During the analysis, all the surface gage readings are written to a file called `GAGES.DAT`. Each row in this file corresponds to a time instant. The first column in the file contains the value of the time. The remaining columns contain the readings of the surface gages. There are four columns of data for each gage. The first column for a gage contains the critical maximum principal normal stress (s_1) over its search range. The second column contains the value of the critical minimum principal normal stress (s_3). The third column contains the critical maximum shear stress (τ_{max}), and the fourth column contains the critical Von Mises' shear stress (s_{vm}). The columns are separated by tabs.

6.2 Finite element probes

Finite Element Probes can be used to output stresses at a particular point when its element number and local coordinates are known. The Element numbering used in the gear tooth finite element meshes is shown in Appendix A. Figure 6.2 shows the finite element probe input menu. The BODY parameter selects the particular body or component to be probed. Each body can have many finite element meshes. The MESH parameter selects which finite element mesh should be probed. There may be many copies or instances of the finite element mesh. Each copy is given an instance number. In the case of a gear tooth mesh, this instance number is the same as the tooth number. The TOOTH parameter selects the instance number. The ELEM parameter selects the finite element number within the mesh. The XI, ETA and ZETA values are the local coordinates within the finite element. XI, ETA and ZETA vary between -1 and $+1$ over the element. Appendix A shows the orientation of the local coordinate axes for each finite element in the various mesh templates.

The COMPONENT parameter selects which stress component should be measured by the probe. Available options are Maximum principal normal stress (s_1), minimum principal normal stress (s_3), maximum shear stress (τ_{max}), Von Mises' octahedral shear stress (s_{vm}) and the displacement magnitude (u). The data measured by the finite element probes is written to a file called `PROBES.DAT`. The data file has a row for each time instant. The first column contains the value of time. Each subsequent column contains the readout of an individual probe.

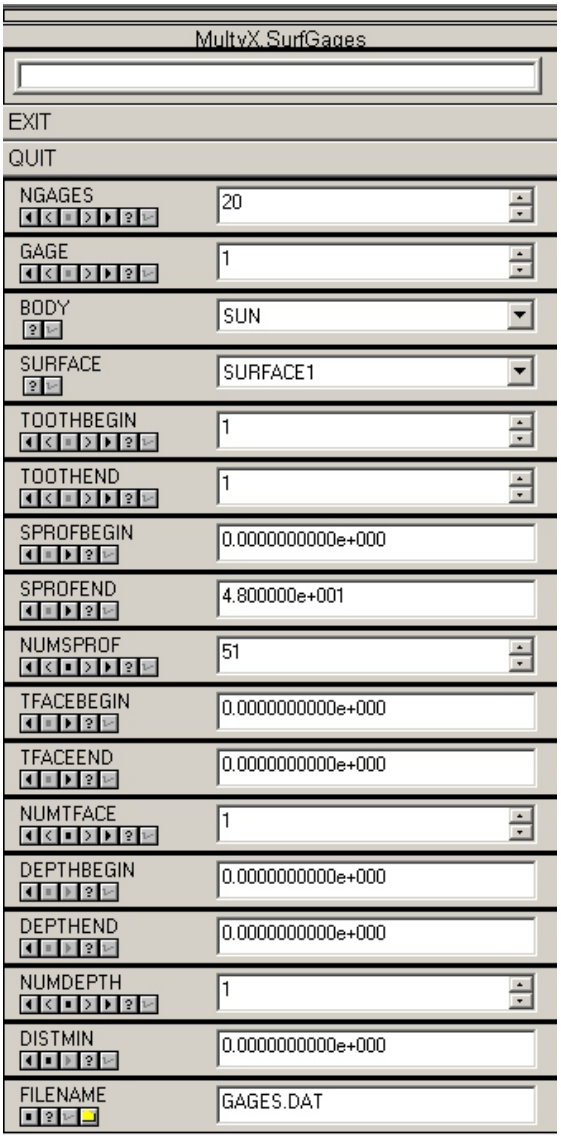


Figure 6.1: The surface gage menu

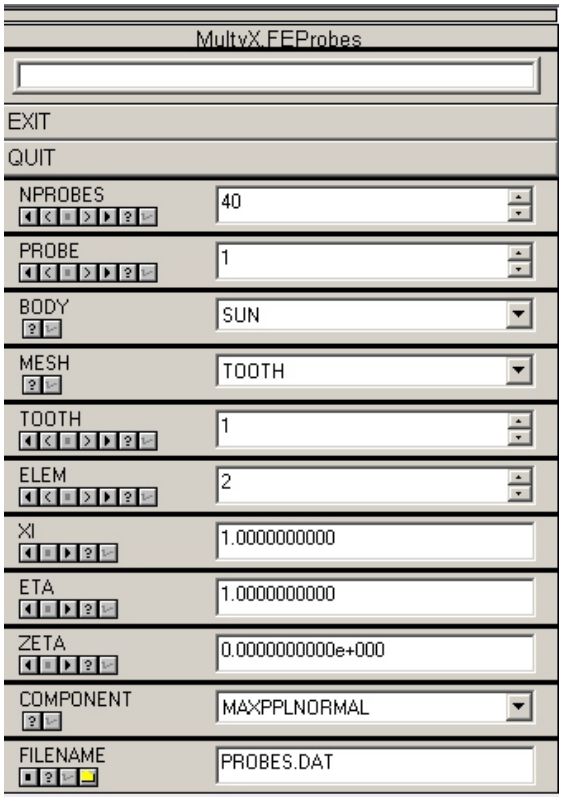


Figure 6.2: The finite element probe menu

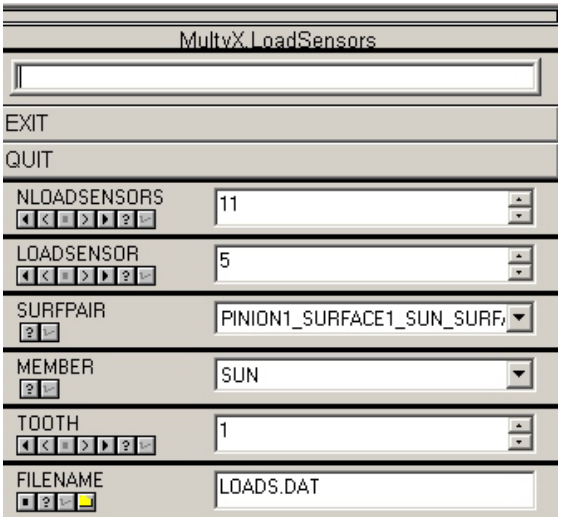


Figure 6.3: The load sensor menu

6.3 Load sensors

Load sensors are used to measure the contact loads generated at the contact surfaces. Figure 6.3 shows the load sensor menu used to set up the sensors. The SURFPAIR item selects the contact surface pair for which the contact load is of interest. Each surface pairing has two contacting members or bodies. The MEMBER parameter selects one of these two bodies, and the TOOTH item selects the individual surface instance number within that body. The outputs of all the sensors are put into a file called `LOADS.DAT`. This file has one row for each instant of time. The first column contains the time. Each subsequent column contains the reading of one load sensor.

6.4 The setup menu

Figure 6.4 shows the analysis setup menu accessed by using the SETUP command at the main menu. The parameters SEPTOL, NPROFDIVS, NFACEDIVS and DSPROF are the grid specification parameters. The variable SEPTOL controls the maximum separation between mating surfaces. Surfaces separated by more than this distance are not considered in the contact analysis. NPROFDIVS parameter contols the number of contact cells that will be used to cover the contact zone between the gear teeth. DSPROFSUN and DSPROFRING control the width of the contact zone in local surface units at the pinion-sun and pinion-ring interfaces respectively. The initial state of the system can be specified as the undeformed state by enabling the ZEROINITIAL flag. The time at which to start the analysis is specified in the INITIALTIME box. If the ZEROINITIAL flag is not checked, then a restart file has to be specified, from which the deformed state and the value of time will be loaded. The analysis time is divided into a user-specified number NRANGES of time ranges. The time step DELTATIME, solution method SOLMETHOD and the number of time steps NTIMESTEPS can be specified separately for each time range.

It is possible to control the operating speed in each time range by specifying a speed factor at the beginning of the range. A speed factor of 1.0 implies that the system is at its nominal speed. The speed factor at the end of a time range is the same as the speed factor at the beginning of the next time range. The speed at the end of the last range is always assumed 1.0. The speed is assumed to vary as a linear function of time within a time range.

The torque in a time range can be controlled by setting the STARTTORQUEFACTOR and ENDTORQUEFACTOR for each range. Again, a factor of 1.0 means that the system is operating at its nominal torque. The torque is assumed to vary as a linear function of time within a time range.

The SAVEPERIODICALLY option saves the state of the system in a restart file after every NSTEPSSAVE number of steps. The state is saved in the restart file named in the SAVEFILENAME box. This restart file can be used to restart another analysis. The OUTPUTRESTART option saves the state of the system in a restart file at the end of the analysis. The file named in the OUTPUTFILENAME box is used. This file can also be used to start a subsequent analysis.

Finally a finite element post-processing data file can be emitted once every NSTEPSWRITE number of time steps by enabling the POSTPROCWRITE option. The file used is selected in the POSTFILENAME box. The post-processing file can be used subsequently to make drawings and stress contour diagrams of the deformed system.

6.5 Other output files

Several tabular output files are created during the analysis.

The displacements and reaction forces generated by the reference frames of the individual bodies in the system are saved in data files during analysis. These data files are named after the bodies. The file `PINION1RES.DAT` contains the results for the pinion1, `SUNRES.DAT` contains results for the sun. Similarly `CARRIERRES.DAT` contains the results for the carrier. Each data file has one row for each instant of time analyzed. The first column contains the time. The next 6 columns contain the six components of reference frame deflection, u_x, u_y, u_z, θ_x, θ_y, and θ_z. The last 6 columns contain the 6 components of reference frame reaction, F_x, F_y, F_z, M_x, M_y, and M_z.

The deformation and reaction forces generated in each bearing are also saved in data files during analysis. These data files are named after the bearings. For example, the file `PINION1BRGRES.DAT` contains the results for the pinion1 bearing, `SUNBRGRES.DAT` contains results for the sun bearing. And `CARRIERBRGRES.DAT` contains the results for the carrier bearing. Each data file has one row for each instant of time analyzed. The first column contains the time. The next 6 columns contain the six components of bearing deformation, u_x, u_y, u_z, θ_x, θ_y, and θ_z. The last 6 columns contain the 6 components of bearing reaction, F_x, F_y, F_z, M_x, M_y, and M_z.

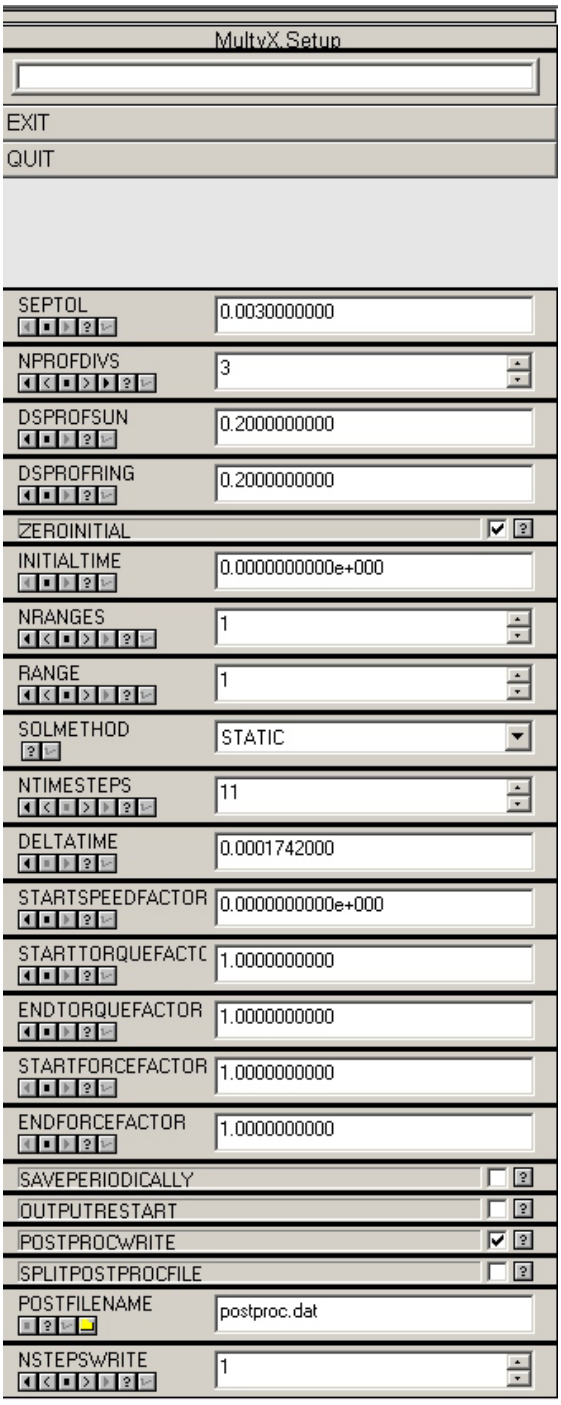


Figure 6.4: The setup menu

Chapter 7

Pre- and Post-processing

The PREPROC command in the main menu leads to the pre-processing menu shown in Figure 7.1. The POSTPROC command leads to the dialog box shown in Figure 7.2, where Multyx asks for the name of the post-processing data file created in the analysis step. When a valid name is entered, the post-processing menu shown in Figure 7.3 comes up.

The pre-processing menu and the post-processing menu are used to make drawings of the system and its components. The CLEAR command clears the graphics screen. The DRAWBODIES draws all the selected bodies using the current view settings. The DRAWBODIES command does not clear the screen before it makes the drawing. In the post-processing menu, the FIRSTPOSN, PREVPOSN, NEXTPOSN, and LASTPOSN commands allow the user to move from one time step saved in the post-processing file to another. Entering a position number directly in the GOTOPOSN box takes the user directly to that time step.

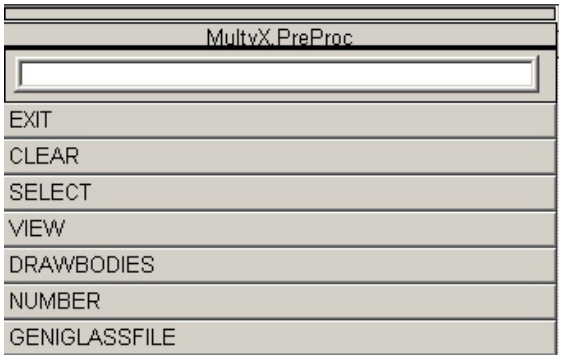


Figure 7.1: The pre-processing menu.

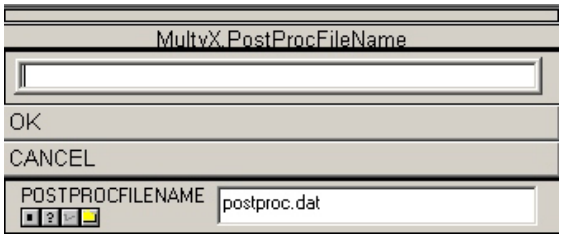


Figure 7.2: The post-processing file name dialog box.

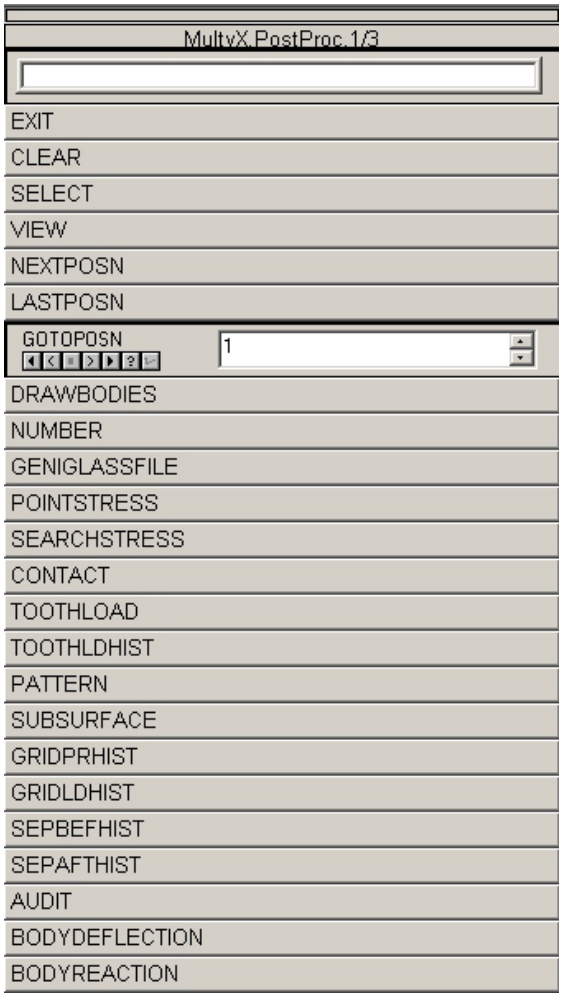


Figure 7.3: The post-processing menu.

7.1 Selecting bodies

The object selection menu which appears when the SELECT command is invoked from the pre- and post-processing menus is shown in Figure 7.4. The objects that should be drawn are selected from this menu.

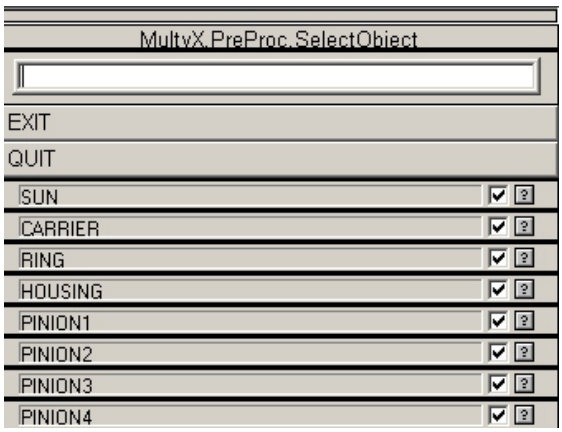


Figure 7.4: The body selection menu.

7.2 View parameters

The VIEW menu controls the appearance of the drawings. In the pre-processing view menu shown in Figure 7.5, the user can enter any value of time into the TIME box. The next drawing will show the system as it would appear at this instant of time. The resolution level controls the degree of detail with which the drawing is made. The ELEMENTS checkbox controls whether or not the individual finite elements should be drawn. The COLORS option controls whether or not the bodies will be filled with color. In pre-processing mode, all bodies are painted Gray. The OUTLINE box controls whether or not an outline drawing of the body will be made.

The view menu in post-processing mode (Figure 7.6) has a few additional parameters. There is a CONTOURS option to draw stress contours. If the COLORS or CONTOURS option is selected, then the menu also asks for the values of the lowest contour level MINSTRESS and the highest contour level MAXSTRESS. The colors used in the drawing are based on the stress level. If the CONTOURS option is checked then the command DELTASTRESS will appear in the menu. You can specify the spacing between the stress contours using this parameter. If the LOADS option is selected, then the contact loads acting on the components will be drawn using the scale factor entered in the LOADSCALE box (Figure 7.7)

If the LOADS option is not checked (Figure 7.6), then an additional box EXAGGERATION appears where an exaggeration factor can by entered for deformed geometry plots. An exaggeration factor of 0.0 will draw the bodies in their undeformed state.

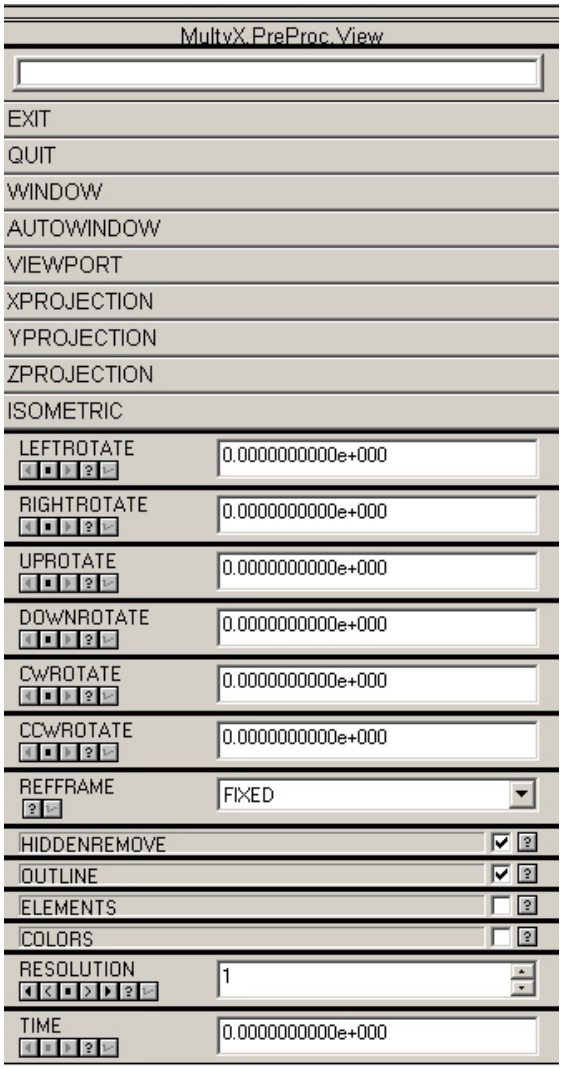


Figure 7.5: The view menu in pre-processing mode

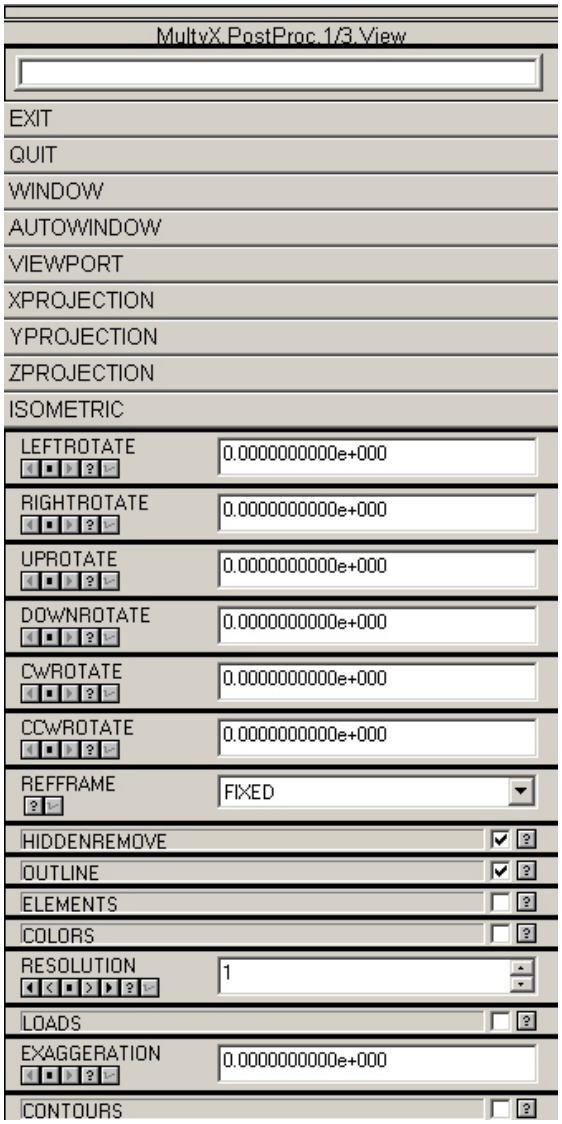


Figure 7.6: The view menu in post-processing mode.

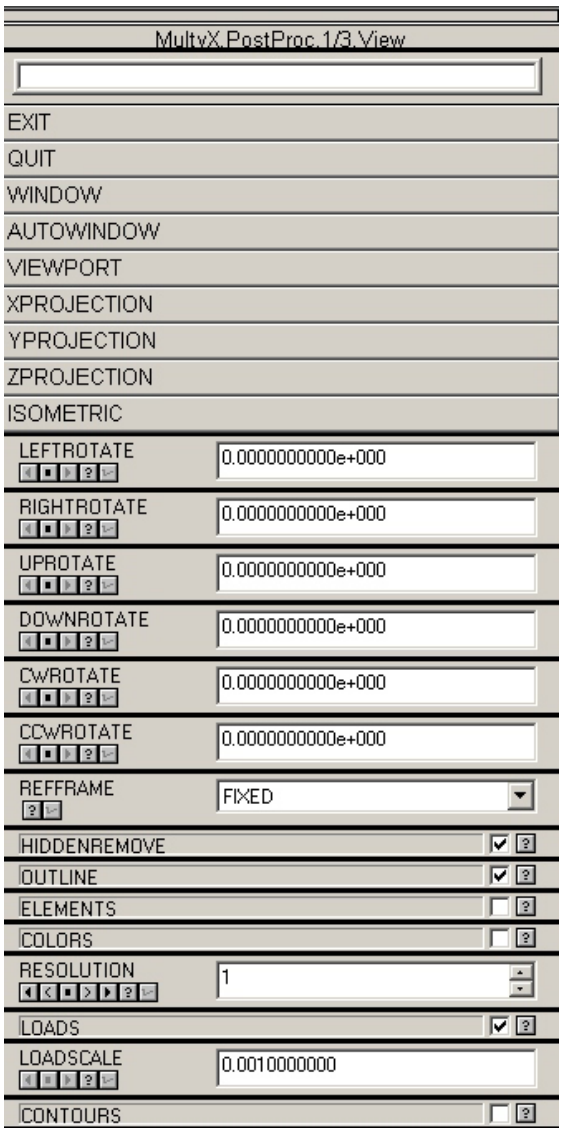


Figure 7.7: The view menu in post-processing mode with the LOADS option enabled.

7.3 The DRAWBODIES command

After an appropriate view and objects have been selected, the DRAWBODIES command in the pre- and post-processing menus (Figures 7.1 and 7.3) will generate a drawing. Figures 7.8 and 7.9 show examples of drawings generated by Multyx in the post-processing mode. Figure 7.10 shows a drawing made in the post-processing mode using the exaggeration factor.

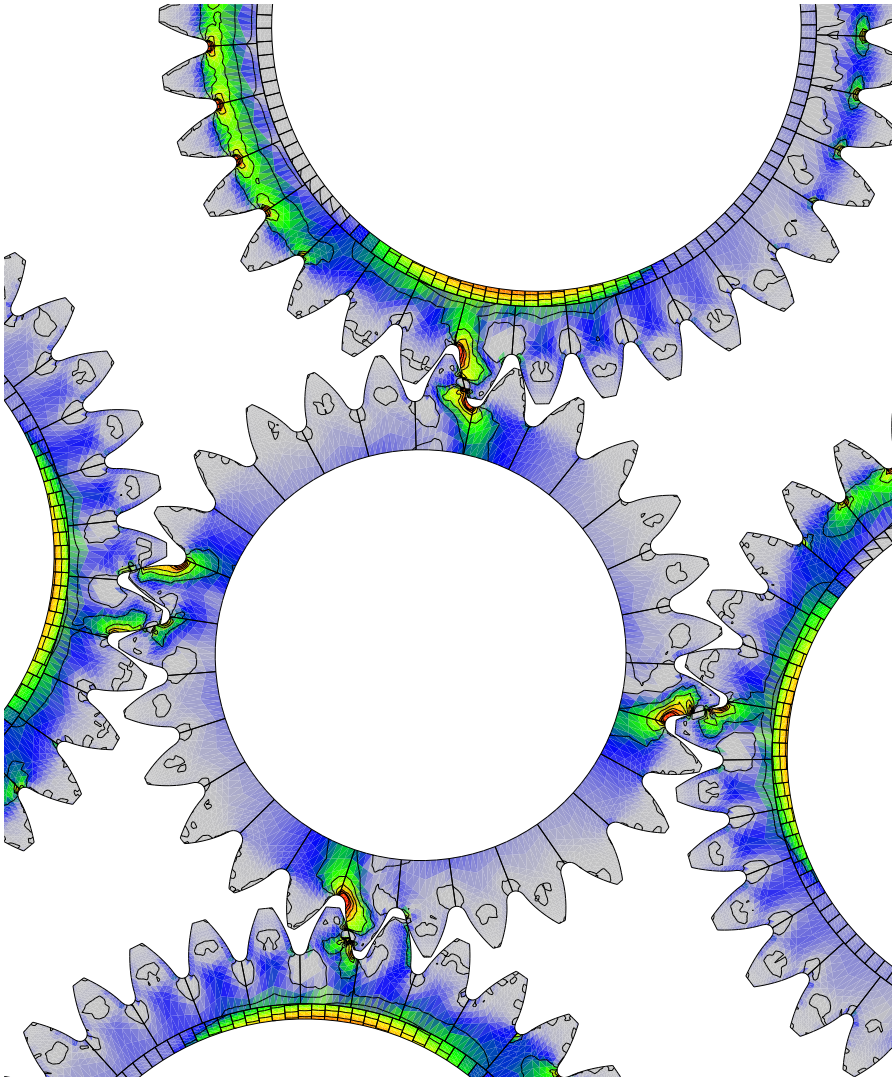


Figure 7.8: An example of a drawing made in the post-processing mode.

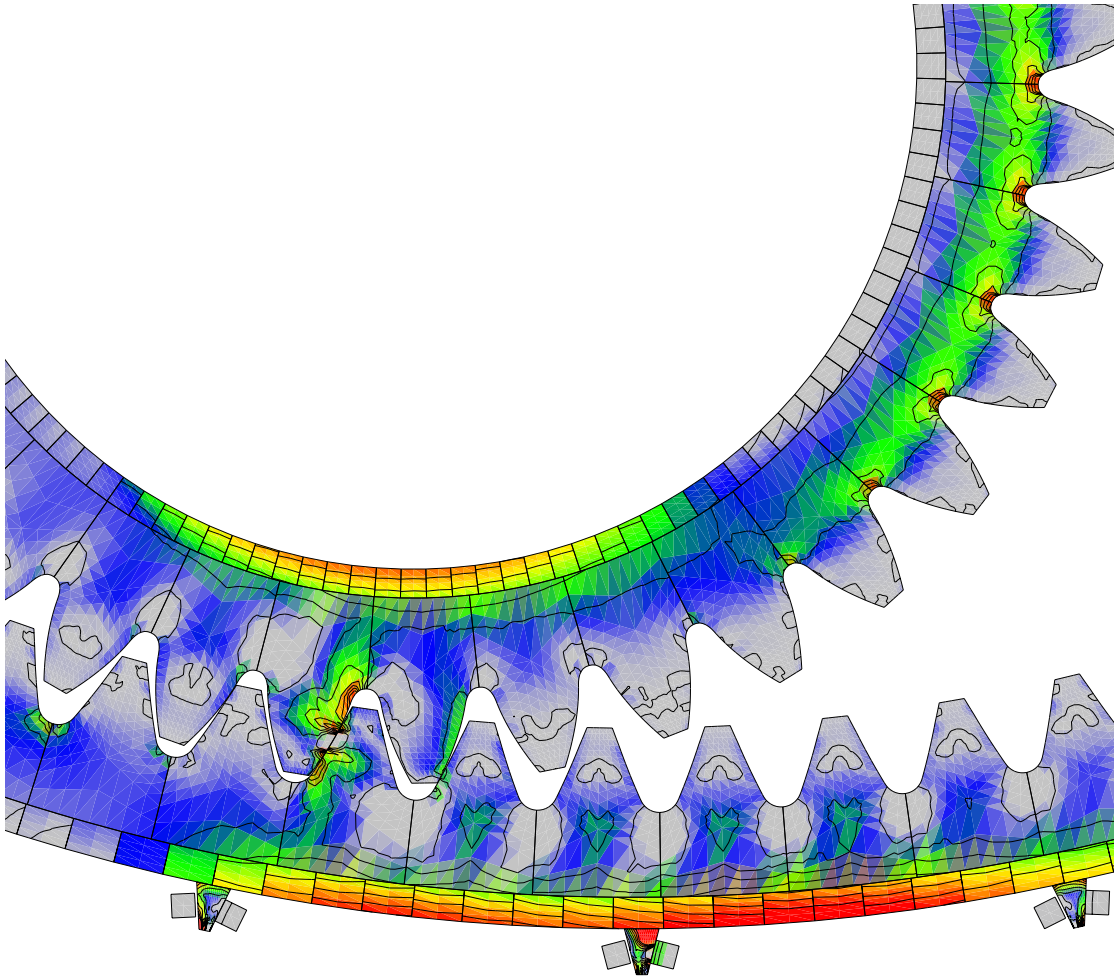


Figure 7.9: An example of a drawing made in the post-processing mode.

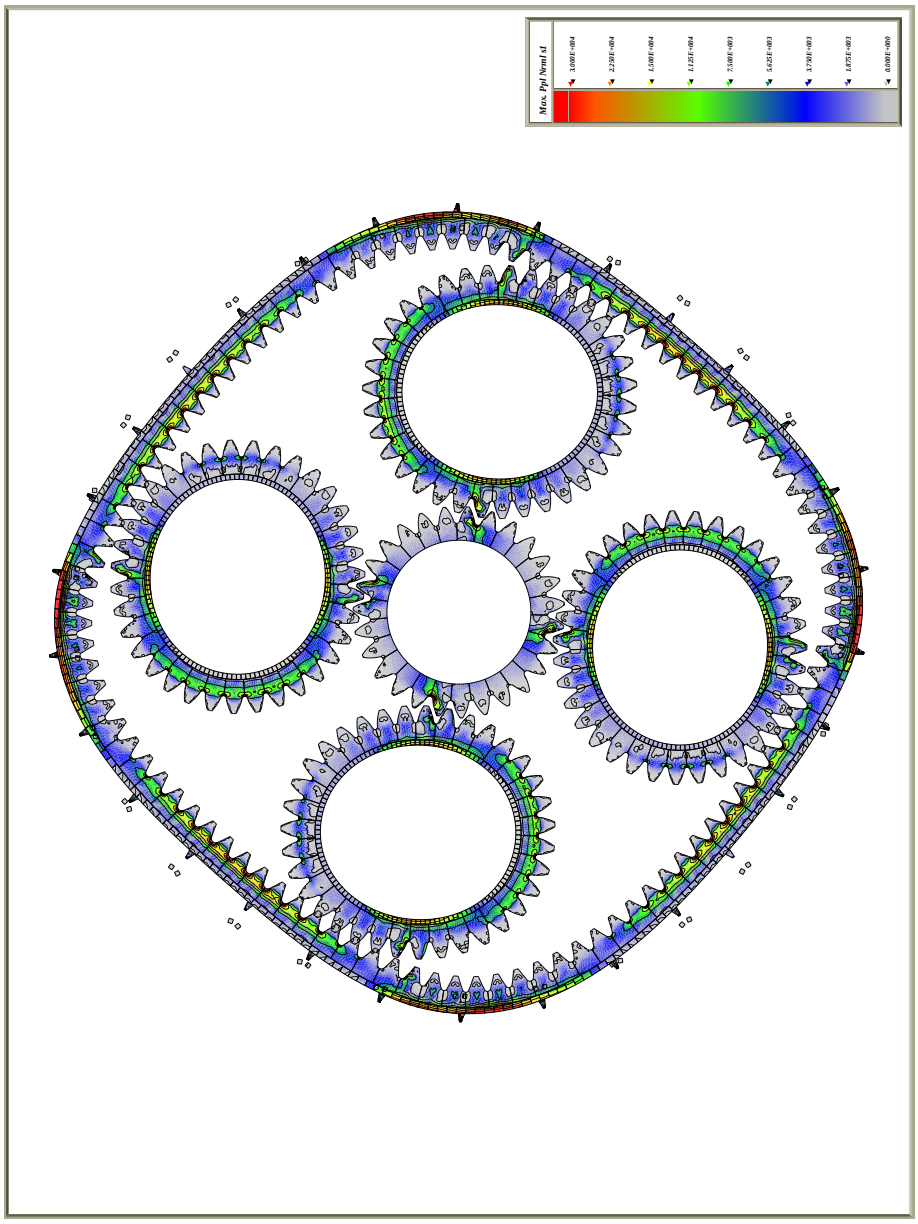


Figure 7.10: An example of a drawing made in the post-processing mode using the exaggeration command.

7.4 The NUMBER command

The NUMBER command in the pre- and post-processing menus (Figures 7.1 and 7.3) lead to the numbering menu shown in Figure 7.11. This menu is used to generate tooth and surface numbering, as shown in Figure 7.12.

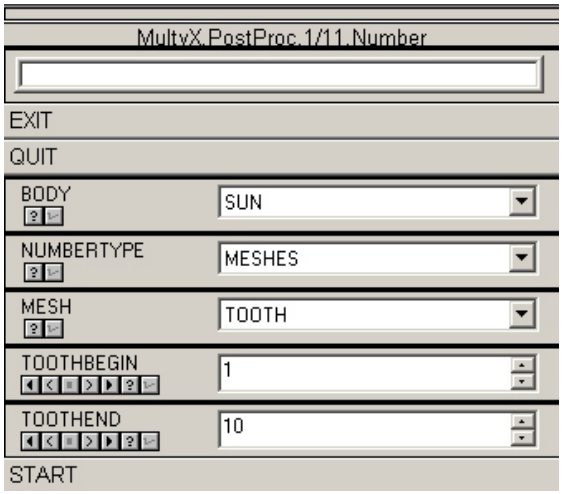


Figure 7.11: The NUMBER menu.

7.5 The TOOTHLOAD command

The TOOTHLOAD command in the post-processing menu (Figure 7.3) leads to the menu shown in Figure 7.13. This menu is used to generate a graph of tooth load vs. time. The SURFACE-PAIR item selects the contact surface pair for which the load is of interest. Each surface pair has two contacting members or bodies. The MEMBER parameter selects one of these two bodies, and the TOOTHBEGIN and TOOTHEND items select a range of instance numbers (or tooth numbers) within that body. If TOOTHBEGIN is greater than TOOTHEND, then the range wraps around the last tooth of the surface. This means that if suppose the model has 27 teeth and you specify the TOOTHBEGIN parameter as 25 and the TOOTHEND parameter as 3 then it will select the range as 25,26,27,1,2 and 3. This range must contain 7 teeth or less.

BEGINSTEP and ENDSTEP are used to select a range of time steps for which results have been stored in the post-processing file. Figure 7.14 shows a graph of tooth load vs. time generated by the TOOTHLOAD command.

The OUTPUTFILENAME item is used to write the tooth load data into an ASCII file. The name of the ASCII file is entered into the item OUTPUTFILENAME. If the APPEND box is checked, and if this file already exists, then the data is appended at the end of the file. Otherwise a new file is created.

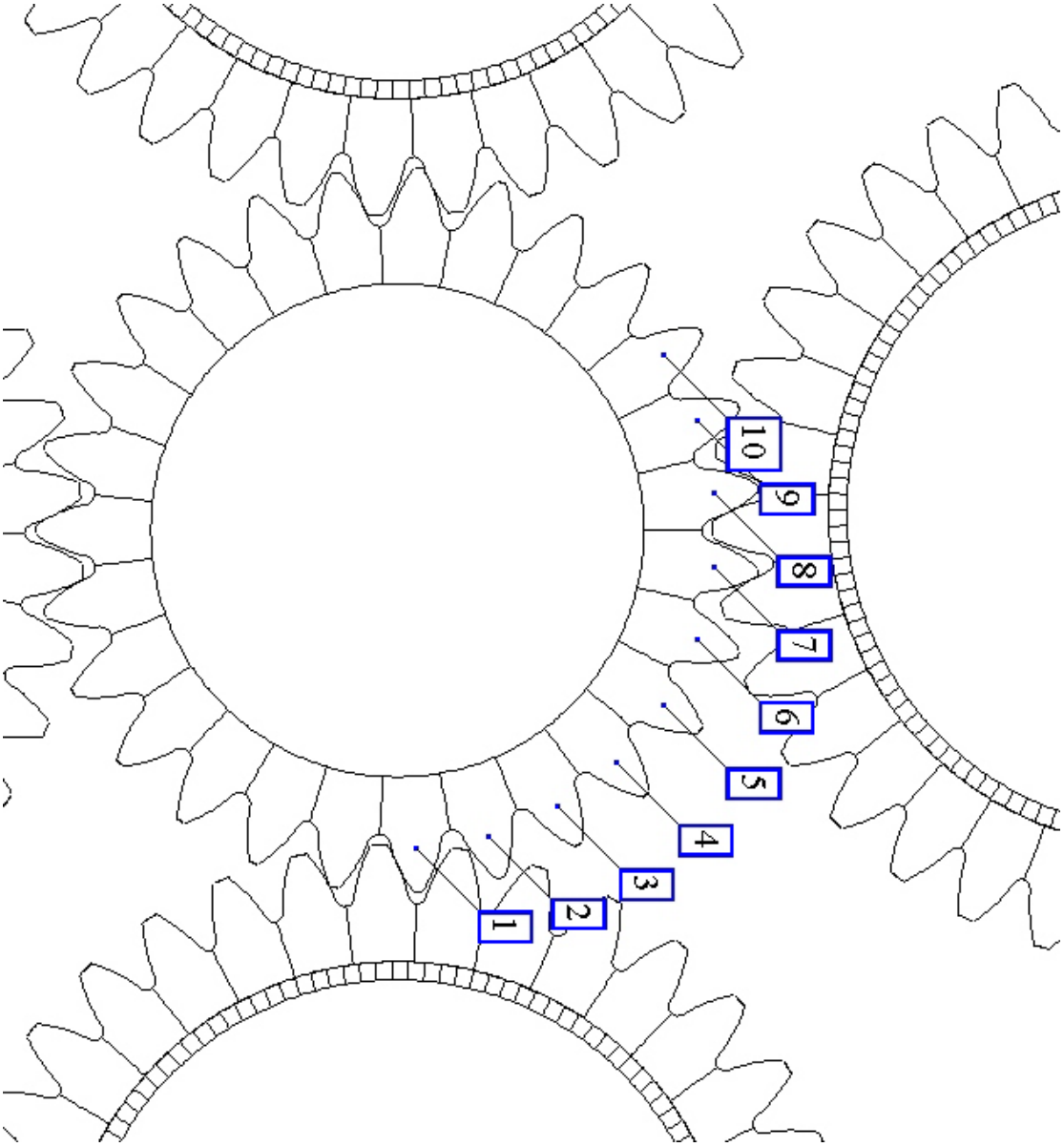


Figure 7.12: Tooth numbering superimposed on a sun gear drawing, using the NUMBER command.

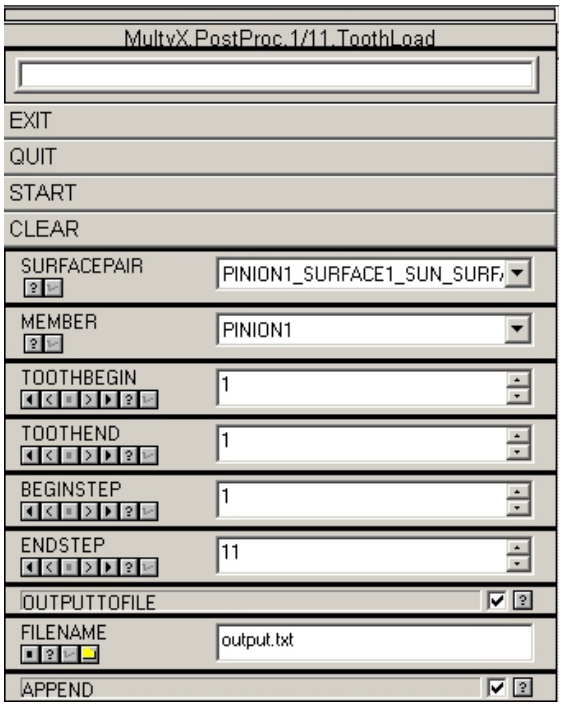


Figure 7.13: The TOOTHLOAD menu.

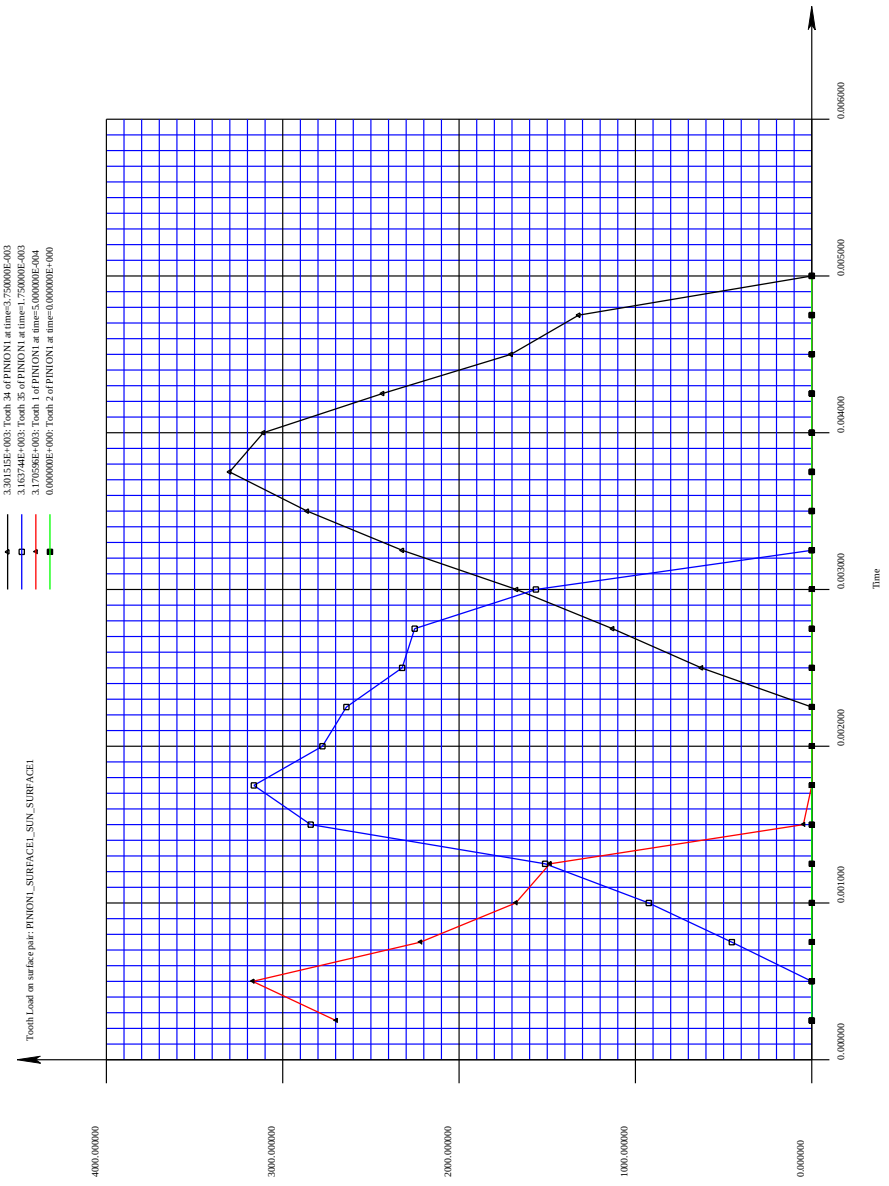


Figure 7.14: The toothload vs. time graph generated by the TOOTHLOAD menu

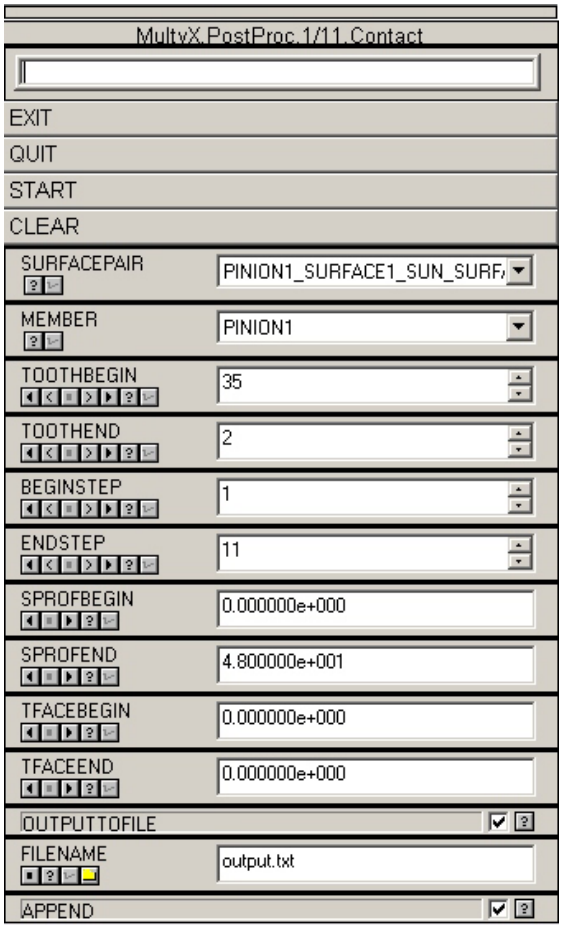


Figure 7.15: The CONTACT menu.

7.6 The CONTACT command

The CONTACT command in the post-processing menu (Figure 7.3) leads to the menu shown in Figure 7.15. This menu is used to generate a graph of contact pressure vs. time.

The SURFACEPAIR item selects the contact surface pair for which the pressure is of interest. Each surface pair has two contacting members or bodies. The MEMBER parameter selects one of these two bodies, and the TOOTHBEGIN and TOOTHEND items select a range of instance numbers (or tooth numbers) within that body. If TOOTHBEGIN is greater than TOOTHEND, then the range wraps around the last tooth of the surface. This range must contain 7 teeth or less. The items SPROFBEGIN, SPROFEND, TFACEBEGIN and TFACEEND are used to restrict the search to a part of the contact surface. Contact occurring outside this range is not considered for display in this graph. You dont need to enter any value for TFACEBEGIN and TFACEEND for a 2D planetary gear set. Figure 7.16 shows a graph of tooth contact pressure vs. time over the surface of the pinion tooth.

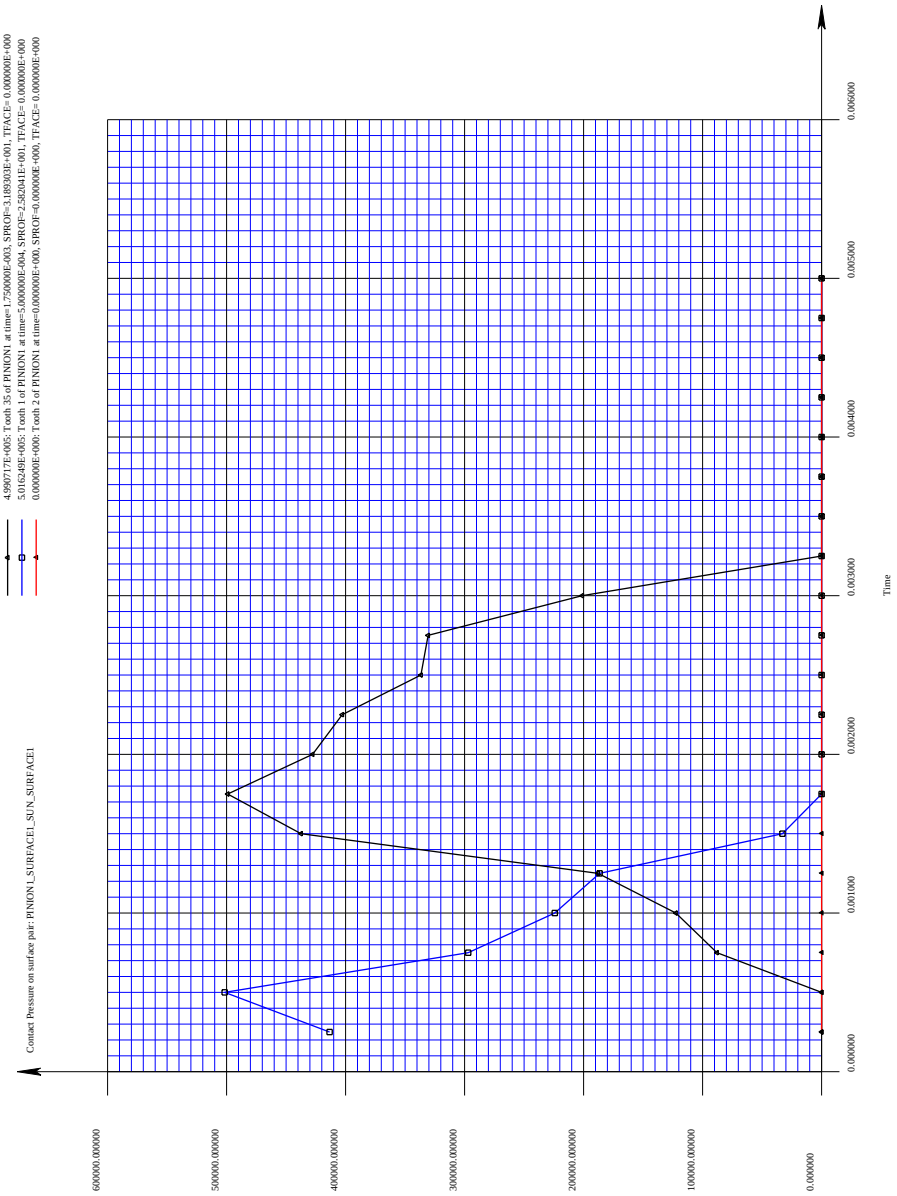


Figure 7.16: The tooth contact pressure vs. time graph generated by the CONTACT menu.

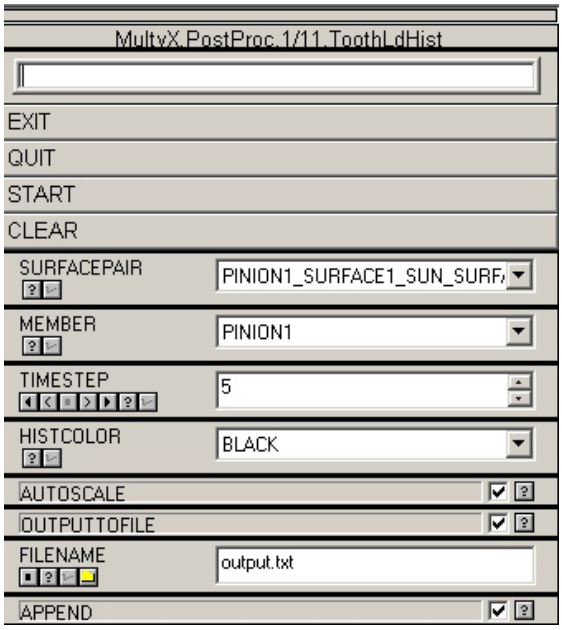


Figure 7.17: The TOOTHLDHIST menu.

7.7 The TOOTHLDHIST command

The TOOTHLDHIST command in the post-processing menu (Figure 7.3) leads to the menu shown in Figure 7.17. This menu is used to generate a histogram of tooth loads at the different teeth in the pinion or gear at a particular time step. The SURFACEPAIR item selects the surface pair, and the MEMBER parameter selects one of the two bodies in this pair. The time step number is selected by the TIMESTEP item. If the AUTOSCALE box is checked, then the vertical scale is automatically computed. Otherwise the user can specify a maximum load value to be used for scaling the vertical axis. The color of the histogram is specified in the HISTCOLOR item. An example of a tooth load histogram is shown in Figure 7.18.

7.8 The SUBSURFACE command

The SUBSURFACE command in the post-processing menu (Figure 7.3) leads to the menu shown in Figure 7.19. This menu is used to generate a graph of subsurface stresses vs. depth under the most critical point in the contact zone at a given timestep. The items TOOTHBEGIN and TOOTHEND are used to select a range of surface instances (tooth numbers). There can be at most 7 teeth in this range.

The items DEPTHBEGIN and DEPTHEND define a depth range, and NUMDEPTH specifies the number of points over this range. Very close to the surface, the subsurface stresses have a large error because of the concentrated nature of the load. So DEPTHBEGIN should never be set to zero.

The stress component is selected in the COMPONENT box. Options available are MAXPPLNORMAL (the maximum principal normal stress s_1), MINPPLNORMAL (the minimum principal normal stress s_3), MAXSHEAR (the maximum shear stress τ_{max}) and VONMISES (the Von Mises' octahedral shear stress s_{VM}).

A subsurface vs. depth graph is not possible for a *Planetary2D* gear set.

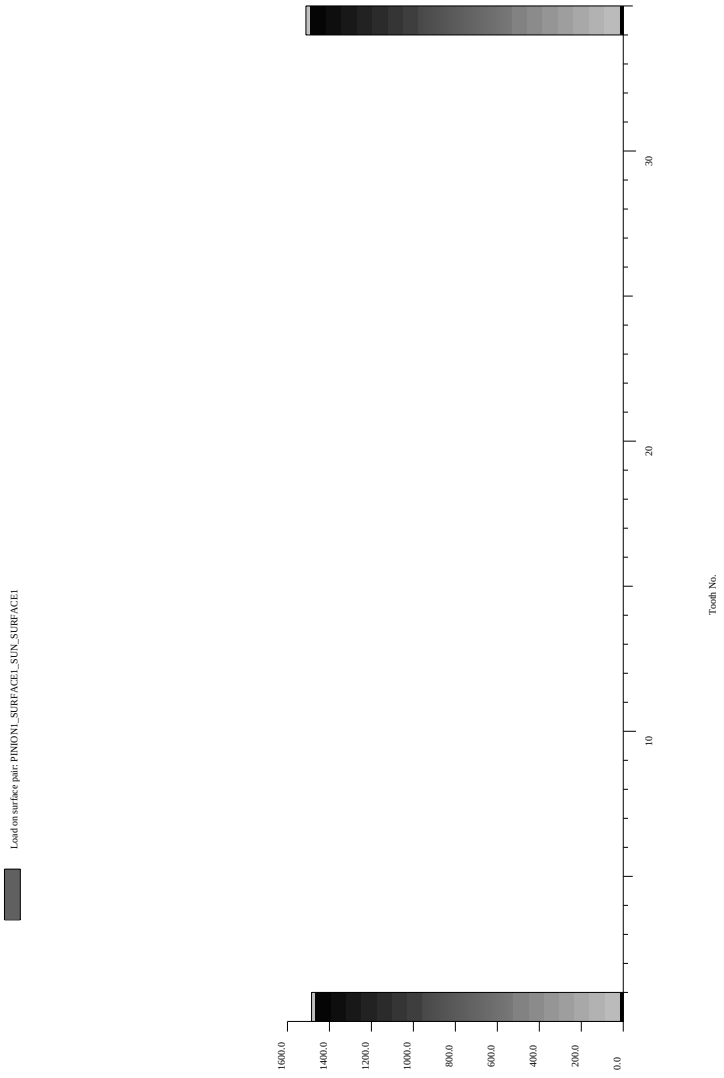


Figure 7.18: The tooth load histogram generated by the TOOTHLDHIST menu.

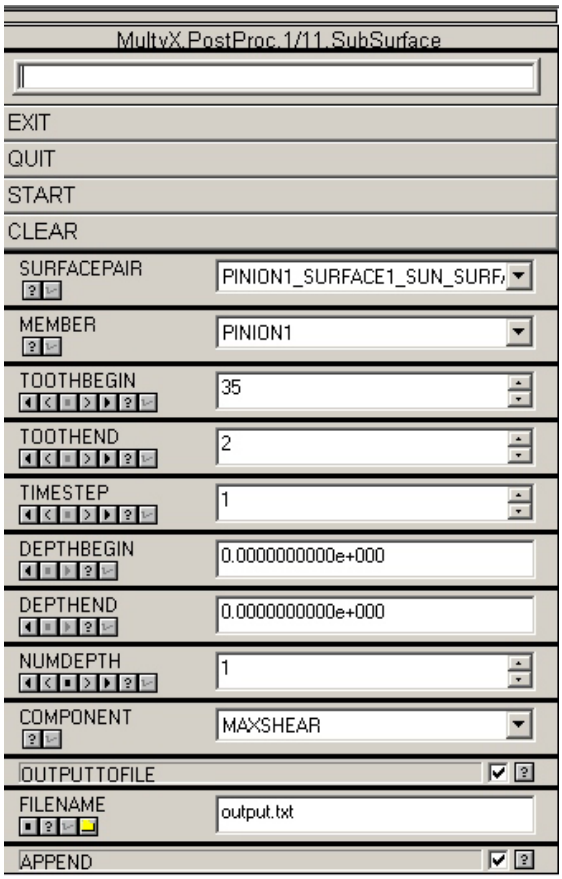


Figure 7.19: The SUBSURFACE menu.

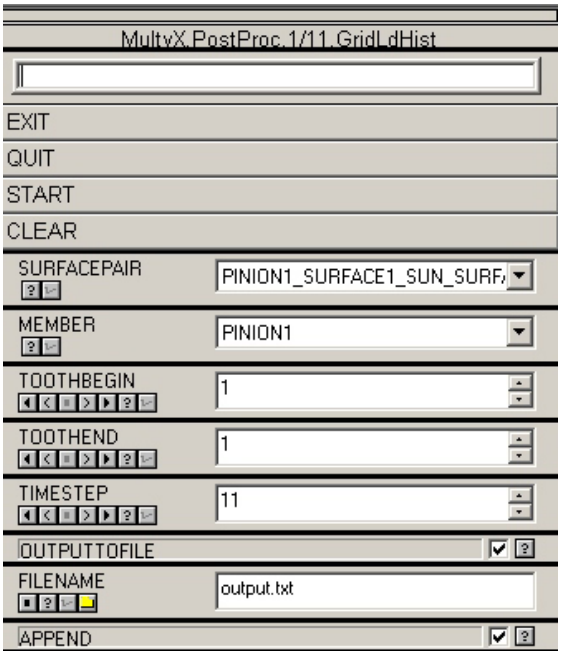


Figure 7.20: The GRIDLDHIST menu.

7.9 The GRIDLDHIST command

The GRIDLDHIST command in the post-processing menu (Figure 7.3) leads to the menu shown in Figure 7.20. This menu is used to generate a histogram of the distribution of contact load over individual contact grid cells. This figure is useful in determining whether the contact grid cell has been properly sized, and whether it has adequate resolution.

The SURFACEPAIR item selects the surface pair, and the MEMBER parameter selects one of the two bodies in this pair. The items TOOTHBEGIN and TOOTHEND are used to select a range of surface instances (tooth numbers). There can be at most 7 teeth in this range. The item TIMESTEP selects a time step number.

Figure 7.21 shows an example of a grid load histogram.

7.10 The GRIDPRHIST command

The GRIDPRHIST command in the post-processing menu (Figure 7.3) leads to the menu shown in Figure 7.22. This menu is used to generate a histogram of the distribution of contact pressure over individual contact grid cells. This command is very similar to the GRIDLDHIST command. The only difference is that it uses contact pressure instead of contact load.

Figure 7.23 shows an example of a grid pressure histogram.

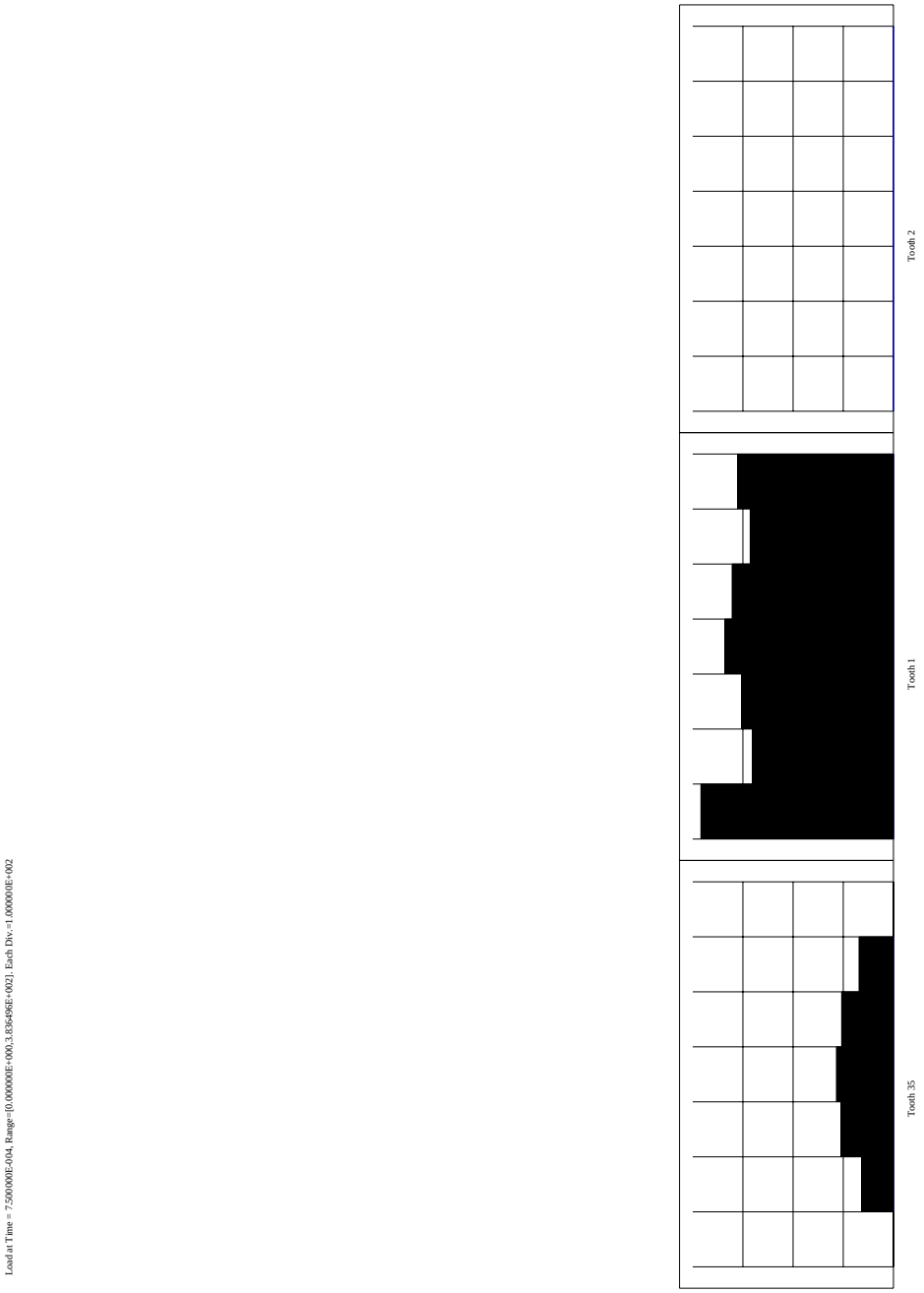


Figure 7.21: The gridload histogram generated by the GRIDLDHIST menu.

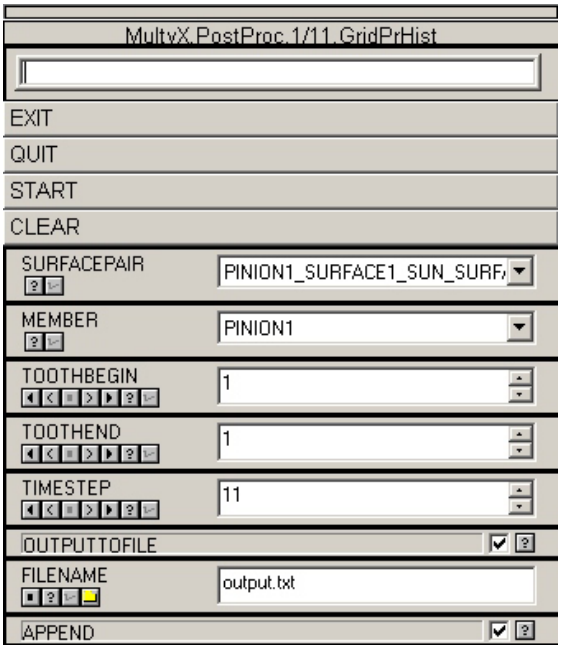


Figure 7.22: The GRIDPRHIST menu.

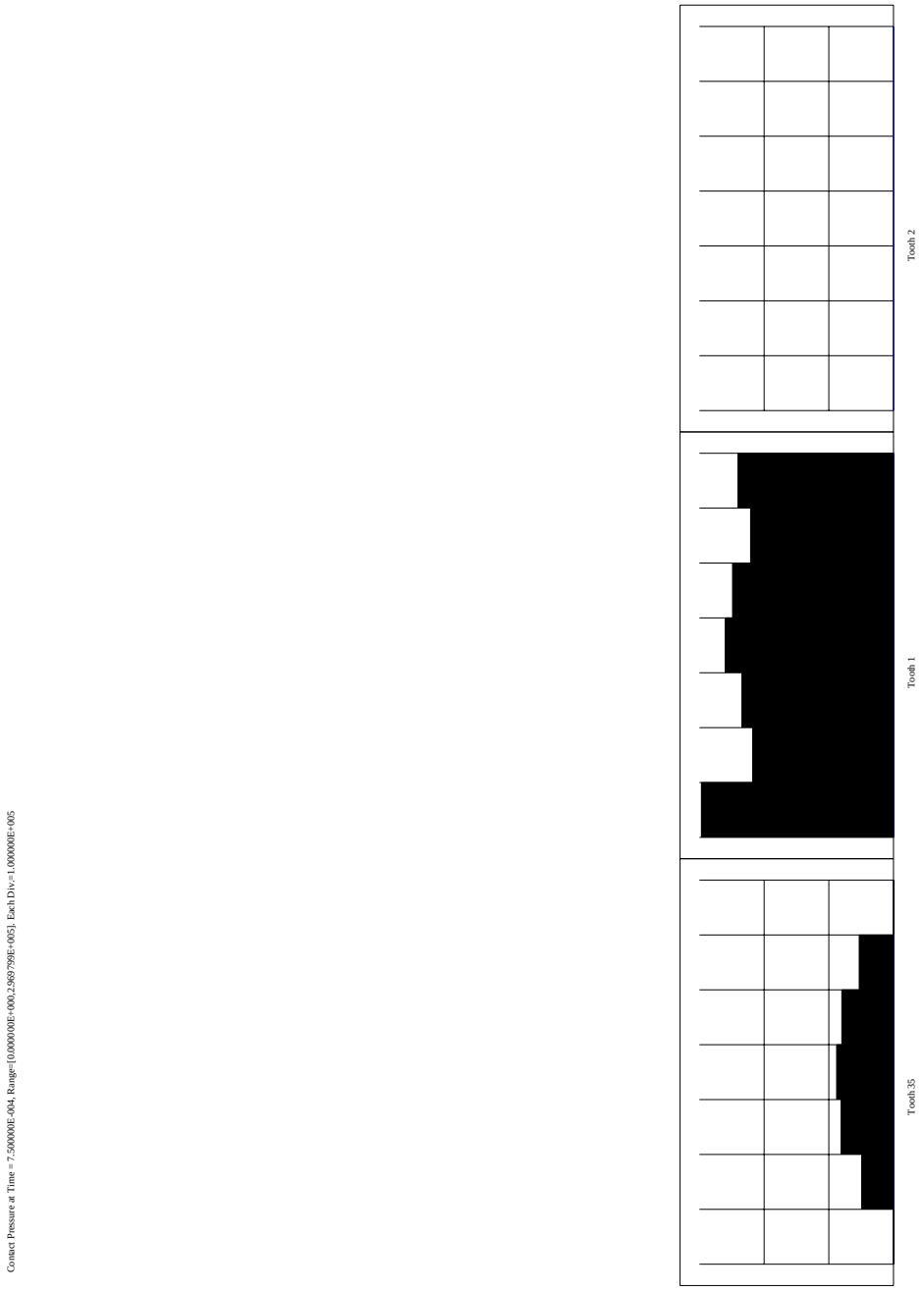


Figure 7.23: The gridload histogram generated by the GRIDPRHIST menu.

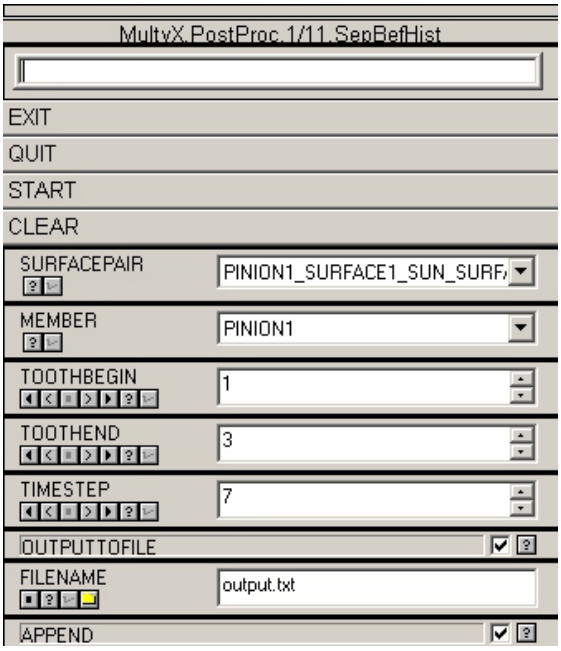


Figure 7.24: The SEPBEFHIST menu.

7.11 The SEPBEFHIST command

The SEPBEFHIST command in the post-processing menu (Figure 7.3) leads to the menu shown in Figure 7.24. This menu is used to generate a histogram of the distribution of normal separation over individual contact grid cells, in the unloaded and undeformed state.

Figure 7.25 shows an example of a histogram of separation in the unloaded state. Negative separation values are possible in this histogram.

7.12 The SEPAFTHIST command

The SEPAFTHIST command in the post-processing menu (Figure 7.3) leads to the menu shown in Figure 7.26. This menu is used to generate a histogram of the distribution of normal separation over individual contact grid cells, in the loaded and deformed state.

Figure 7.27 shows an example of a histogram of separation in the loaded state. These separation values must be either zero or positive.

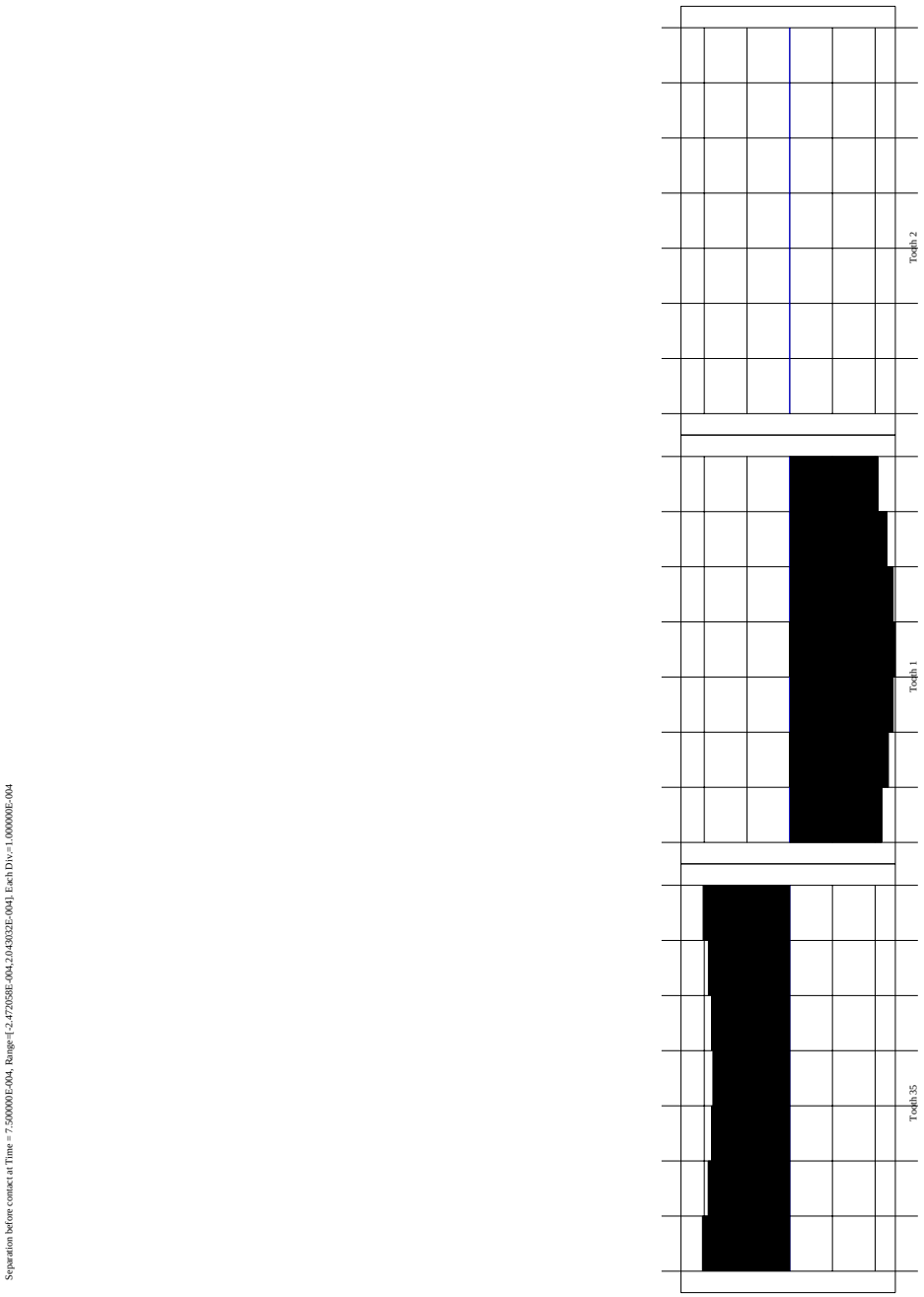

Separation before contact at Time = 7.500000E-004, Range=[-2.472058E-004,2.043032E-004], Each Div.=1.000000E-004
Tooth 2
Tooth 1
Tooth 35

Figure 7.25: The histogram of grid separation before contact, generated by the SEPBEFHIST menu.

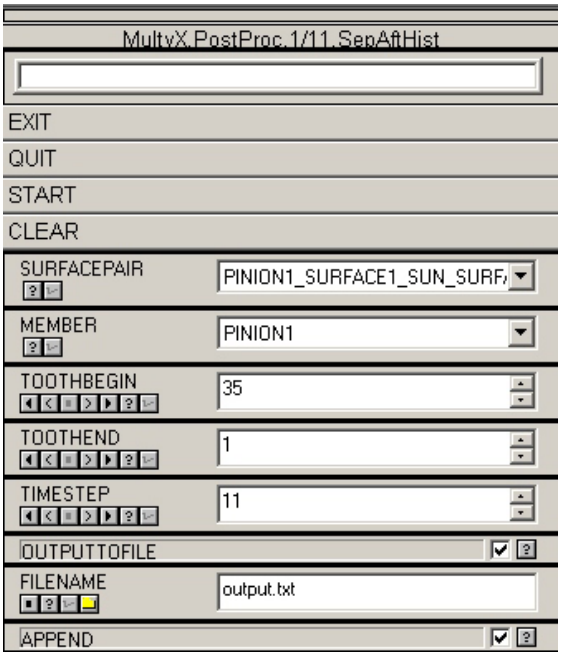


Figure 7.26: The SEPAFTHIST menu.

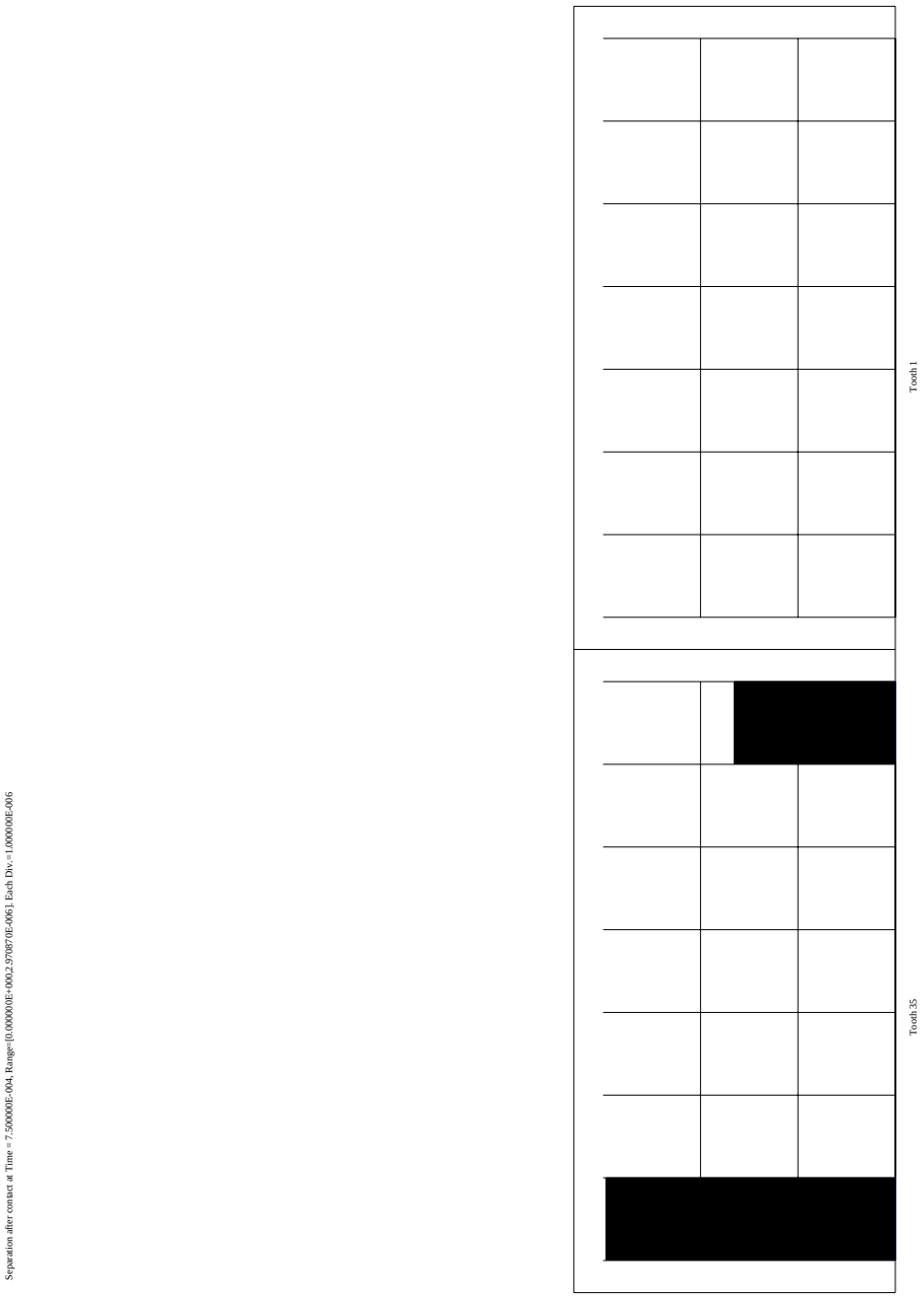


Figure 7.27: The histogram of grid separation after contact, generated by the SEPAFTHIST menu.

7.13 The SEARCHSTRESS command

The SEARCHSTRESS command of the post-processing menu (Figure 7.3) leads to the menu shown in Figure 7.28. This menu is used to locate the most critical stresses in the system.

The COMPONENT box is used to select the stress component of interest. Available choices are MAXPPLSTRESS (the maximum principal normal stress s_1), MINPPLSTRESS (the minimum principal normal stress s_3), MAXSHEAR (the maximum shear stress τ_{max}), and VONMISES (the Von Mises' octahedral shear stress s_{VM}).

Depending on selection in the XAXIS box, the stress can be displayed as a function of time (TIME), profile (SPROF), face (TFACE) or depth (DEPTH).

The stress values are computed over a range of time steps (specified by BEGINSTEP and ENDSTEP), teeth (specified by TOOTHBEGIN and TOOTHEND), location along the profile (specified by SPROFBEGIN, SPROFEND and NUMSPROF), location along the face (specified by TFACEBEGIN, TFACEEND and NUMTFACE), and depth (specified by DEPTHBEGIN, DEPTHEND and NUMDEPTH).

If the number of teeth defined by TOOTHBEGIN and TOOTHEND is 7 or less, and if the SEPTEETH box is checked, then a separate graph is drawn for each tooth. Otherwise a single graph is drawn showing the most critical stress among all the teeth in the range.

File output is controlled by the OUTPUTTOFILE, FILENAME and APPEND items. Figure 7.29 shows stress as a function of time, Figure 7.30 shows stress as a function of profile.

7.14 The POINTSTRESS command

The POINTSTRESS command of the post-processing menu (Figure 7.3) leads to the menu shown in Figure 7.31. This menu is used to track normal stresses in a specific direction at a specific point on a surface.

The surface is selected by specifying the body in the BODY box and a surface in the SURFACE box. A range of teeth with up to 7 teeth is selected through the TOOTHBEGIN and TOOTHEND items. A profile and face location on this surface is specified through the SPROF and TFACE parameters.

The direction is specified by an angle in the item ANGLE. This angle is the angle between the normal direction of interest and the profile direction (if the REFDIRECTION option is SPROF). The angle is measured using the right hand rule about the outward normal to the surface.

The range of time steps is specified by the BEGINSTEP and ENDSTEP items. File output is controlled by the OUTPUTTOFILE, FILENAME and APPEND items.

Figure 7.32 shows an example of the graph generated by this menu.

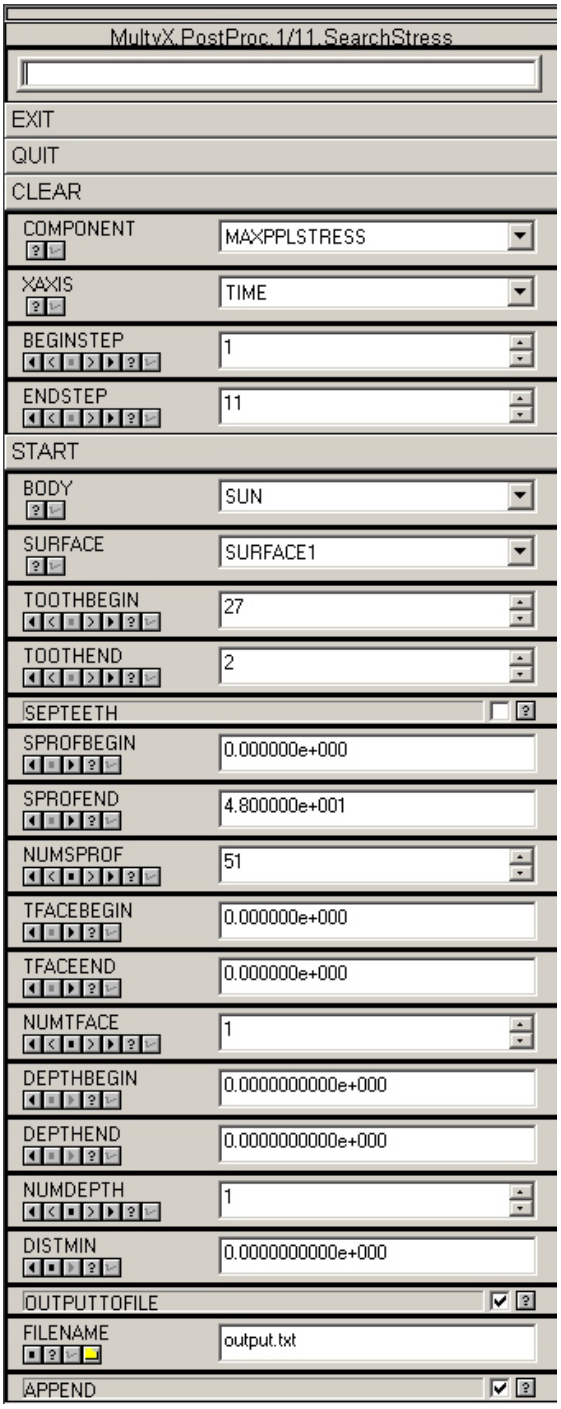


Figure 7.28: The SEARCHSTRESS menu

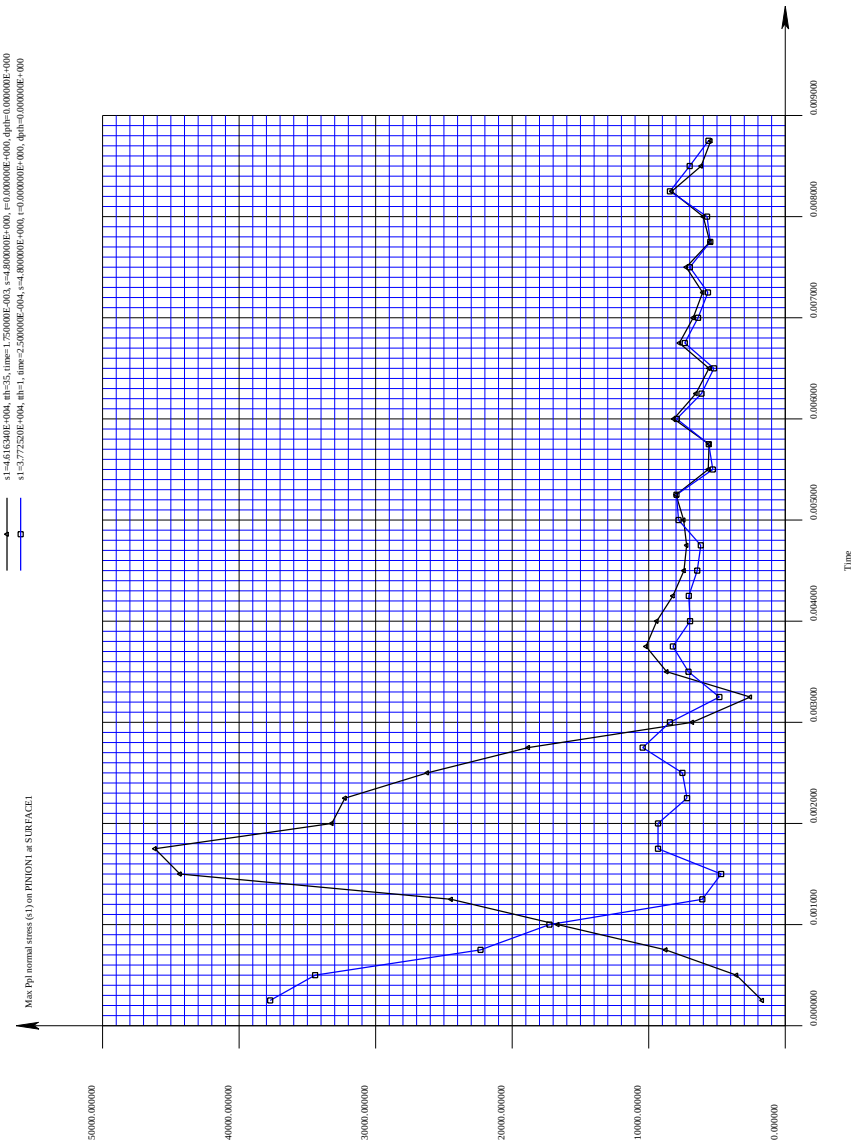


Figure 7.29: The graph of root stress vs. time, generated by the SEARCHSTRESS menu.

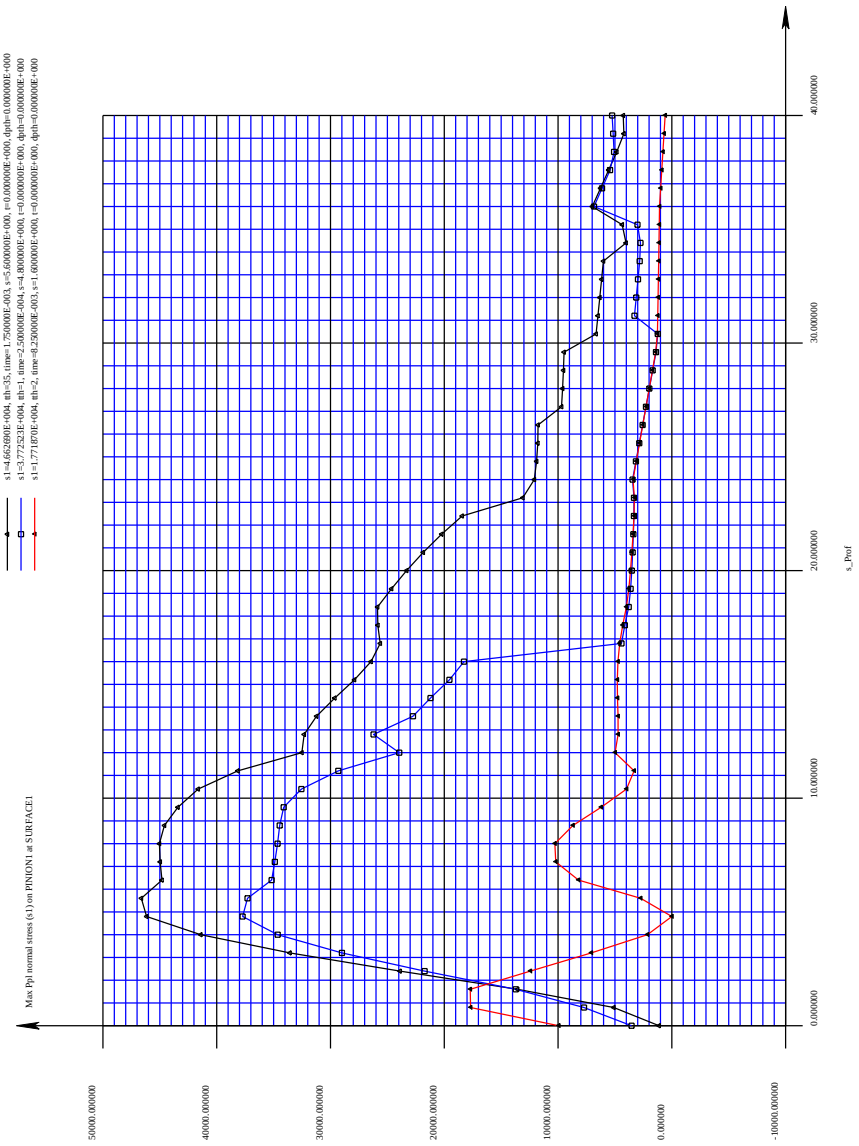


Figure 7.30: The graph of root stress vs. profile, generated by the SEARCHSTRESS menu.

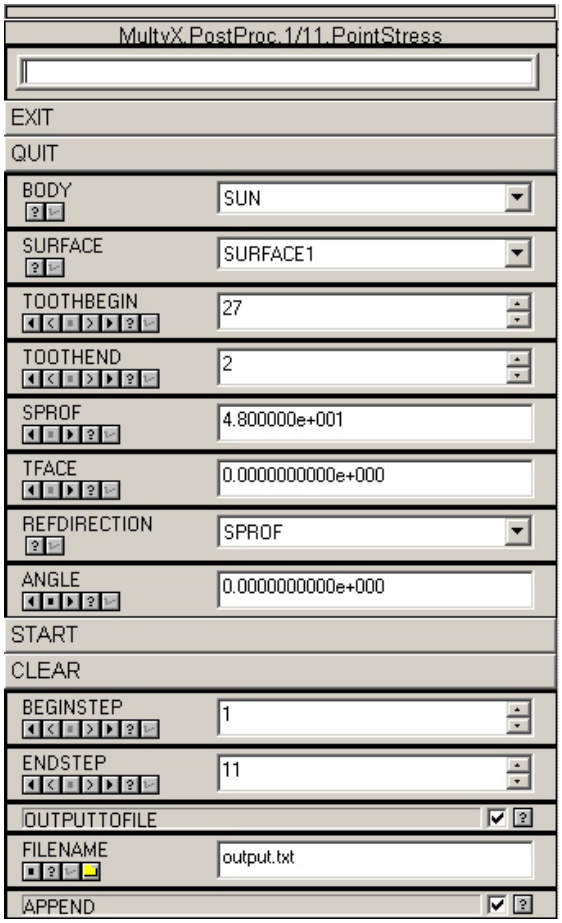


Figure 7.31: The POINTSTRESS menu

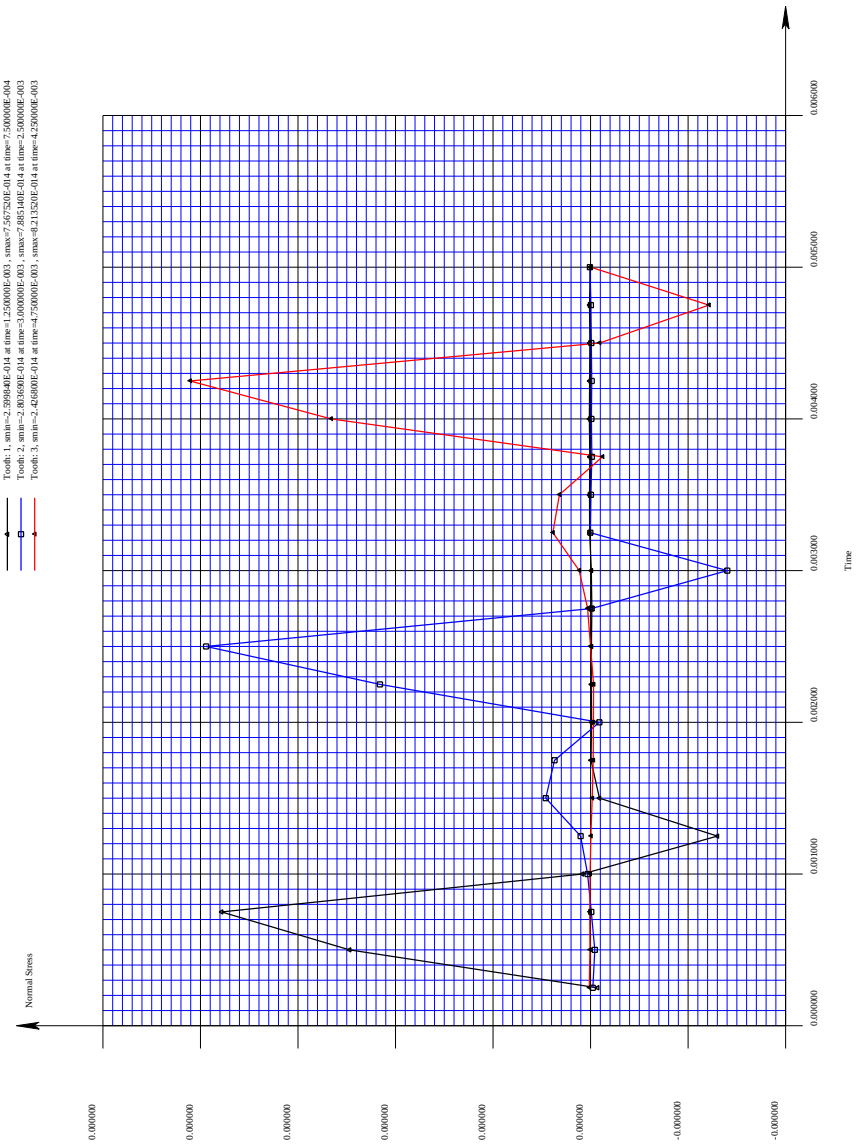


Figure 7.32: The graph of root stress vs. time, generated by the POINTSTRESS menu.

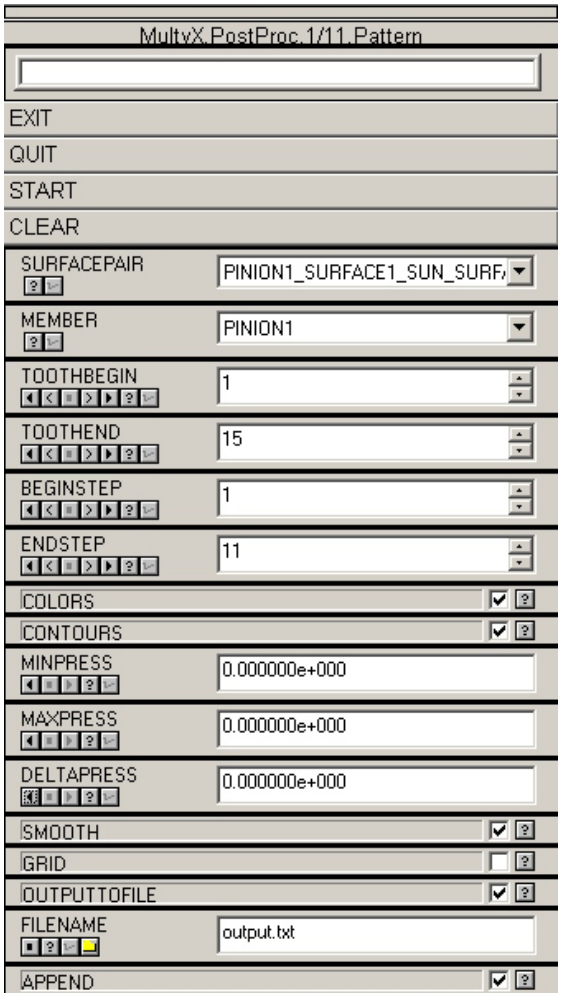


Figure 7.33: The PATTERN menu.

7.15 The PATTERN command

The PATTERN command of the post-processing menu (Figure 7.3) leads to the menu shown in Figure 7.33. This menu is used to create a drawing of the contact pattern on a tooth.

The surface is selected by specifying the body in the BODY box and a surface in the SURFACE box. A range of teeth with up to 7 teeth is selected through the TOOTHBEGIN and TOOTHEND items. The range of time steps is specified by the BEGINSTEP and ENDSTEP items.

The contact pattern can be displayed in color if the COLORS box is checked, or with contour lines if the CONTOURS box is checked. A contact pattern cannot be generated for a *Planetary2D* gear set since it does not have any thickness.

The contact pattern drawing is not three-dimensional. It is a projection of the contact surface in the $r - z$ coordinate plane.

If the SMOOTH box is checked, then the contact pressures will be smoothed by fitting a polynomial surface to the raw data.

7.16 The AUDIT command

Frequently the user needs to obtain the force and moment balance for the individual bodies in the system. The AUDIT command of the post-processing menu (Figure 7.3) generates an equilibrim 'audit' of all the forces and moments acting on each body. Figure 7.34 shows the AUDIT sub-menu. The list of bodies for which this audit is to be generated is selected through a sub-menu accessed through the SELECT button in this menu. The range if time steps is specified in the BEGINSTEP and ENDSTEP boxes.

The START button then displays the audit statement in the Information window. It can also be sent to an ASCII file by using the OUTPUTTOFILE, FILENAME and APPEND boxes. A sample equilibrium audit for the sun is shown below:

```
Time=0.00025
Body no.1:SUN (Origin at:[0,0,0])
================================================================

Contact forces:
    Exerted by:PINION1
    Total  :f [-1099.798249,2462.388408,0],
            mo[0,0,3740.488587]
            m [0,0,3740.488587]
    Exerted by:PINION2
    Total  :f [-2431.622595,-1159.733387,0],
            mo[0,0,3736.616789]
            m [0,0,3736.616789]
    Exerted by:PINION3
    Total  :f [1120.528974,-2489.569011,0],
            mo[0,0,3775.141985]
            m [0,0,3775.141985]
    Exerted by:PINION4
    Total  :f [2474.940612,1191.830242,0],
            mo[0,0,3794.698401]
            m [0,0,3794.698401]
Total contact force=f [64.04874165,4.916252665,0]
                    mo[0,0,15046.94576]
                    m [0,0,15046.94576]


Bearing forces:
SUNBRG:f [-62.36947675,-4.787355666,0],
                       mo[0,0,0]
                       m [0,0,0]
Total bearing force=f [-62.36947675,-4.787355666,0],
                    mo[0,0,0]
                    m [0,0,0]


Total internal force (inertial+press+body):f [3.891078848e-015,8.351416196e-015,0],
                                            mo[0,0,7.478249975e-007]
                                            m [0,0,7.478249975e-007]
Total mass & damping force                 :f [-1.6792649,-0.1288969982,0],
                                            mo[0,0,1153.054238]
                                            m [0,0,1153.054238]
Total contact  force                       :f [64.04874165,4.916252665,0],
                                            mo[0,0,15046.94576]
                                            m [0,0,15046.94576]
Total bearing  force                       :f [-62.36947675,-4.787355666,0],
                                            mo[0,0,0]
                                            m [0,0,0]
Total reaction force                       :f [0,0,0],
                                            mo[0,0,-16200]
                                            m [0,0,-16200]
================================================================================
Residual force (error)                     :f [-5.613287613e-013,1.296740493e-013,0],
                                            mo[0,0,3.637978807e-012]
                                            m [0,0,3.637978807e-012]
```

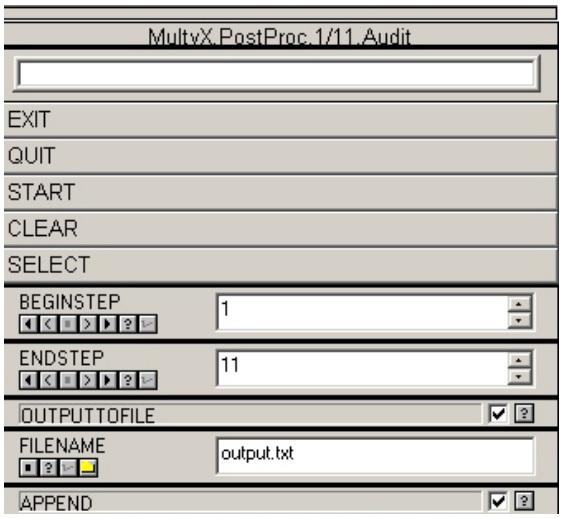


Figure 7.34: The AUDIT menu.

The forces (and moments) are broken down into contact forces, bearing forces, internal forces, mass and damping forces and reaction forces. The reaction forces are the forces exerted by the reference frame constraints.

Two values for the moments are displayed. In the above example, `mo` refers to the moments computed about the origin of the sun gear. `m` stands for the moment computed about the origin of the fixed reference frame. The moments about the fixed reference frame are more useful in comparing the action and reaction acting on different bodies.

Regardless of the origin about which the moments are computed, the X Y and Z components of each force and moment always refer to the fixed reference frame.

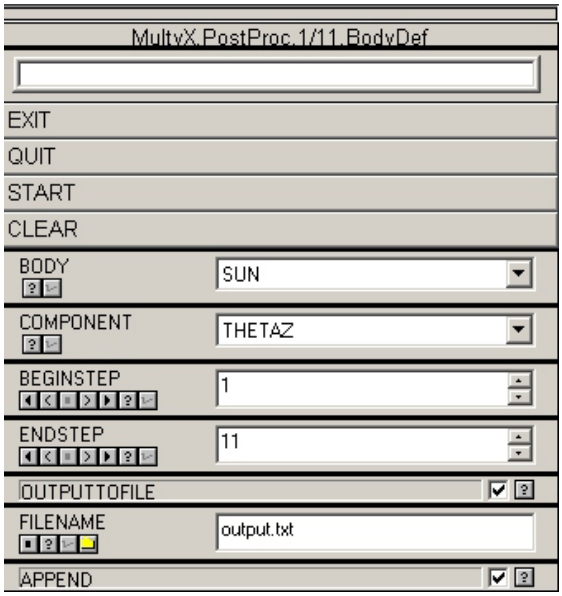


Figure 7.35: The BODYDEFLECTION menu.

7.17 The BODYDEFLECTION command

The BODYDEFLECTION command of the post-processing menu (Figure 7.3) leads to the menu shown in Figure 7.35. This menu is used to generate a graph of a component of the rigid body type motion of a body as a function of time as shown in Figure 7.36. The six components of motion that can be graphed are the 3 translation motions u_x, u_y and u_z, and the three rotation components θ_x, θ_y and θ_z. These components are calculated in the reference frame attached to the body. The rotation components are displayed in Radians.

7.18 The BODYREACTION command

The BODYREACTION command of the post-processing menu (Figure 7.3) leads to the menu shown in Figure 7.37. This menu is used to generate a graph of a component of the body frame reaction as a function of time as shown in Figure 7.38. The six force components that can be graphed are the three forces F_x, F_y and F_z, and the three moments M_x, M_y and M_z. These components are calculated in the reference frame attached to the body. The moments are computed about origin of this reference frame.

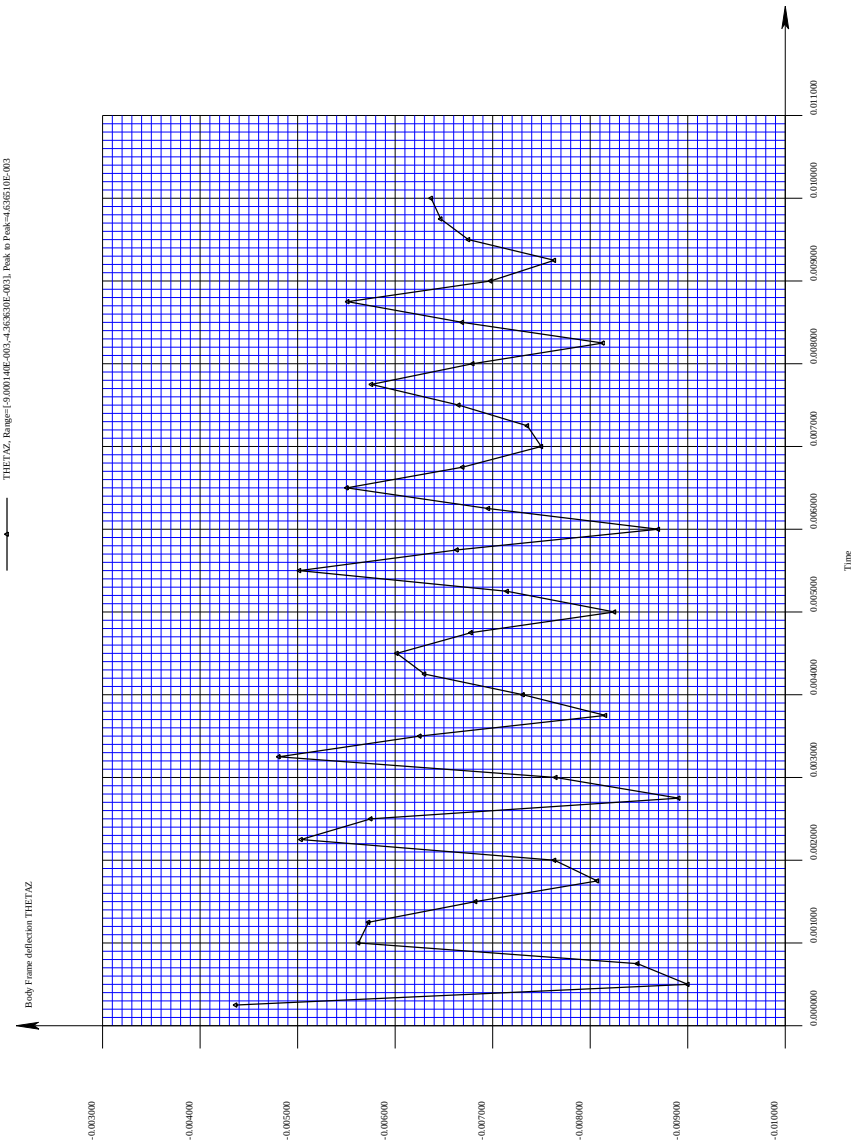


Figure 7.36: The graph generated by the BODYDEFLECTION menu.

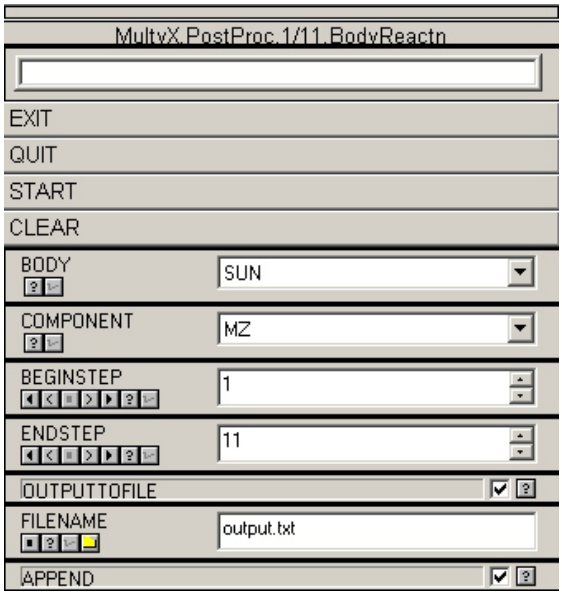


Figure 7.37: The BODYREACTION menu.

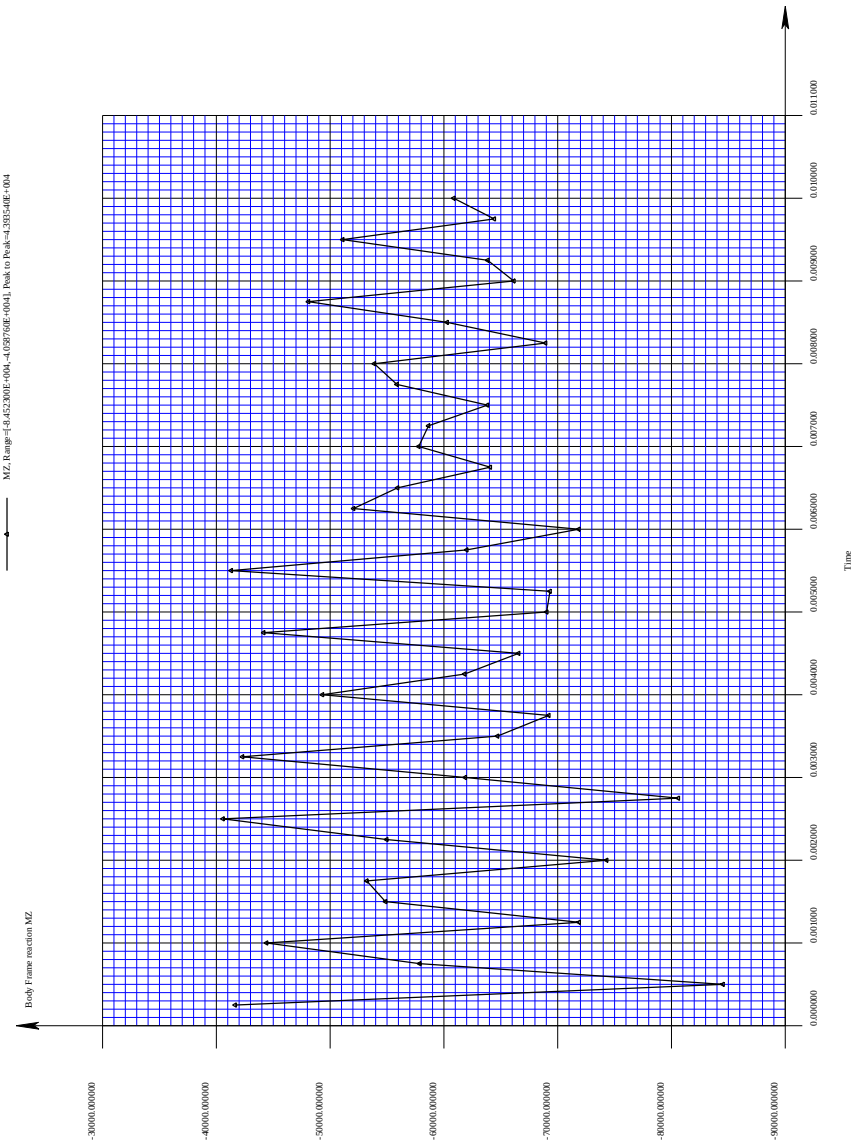


Figure 7.38: The graph generated by the BODYREACTION menu.

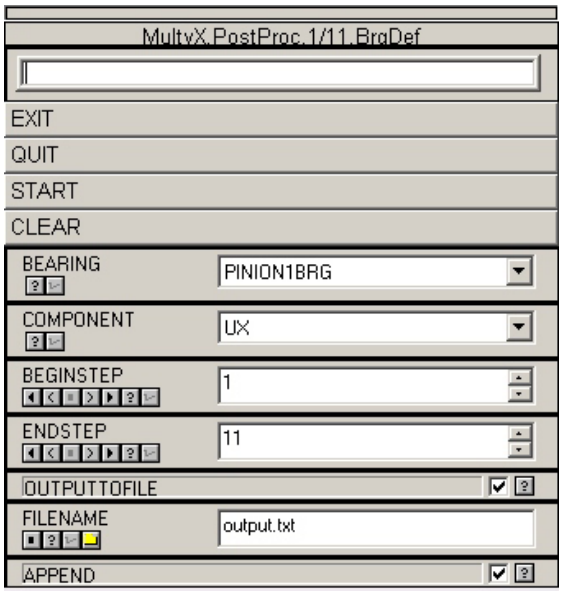


Figure 7.39: The BRGDEFORMN menu.

7.19 The BRGDEFORMN command

The BRGDEFORMN command of the post-processing menu (Figure 7.3) leads to the menu shown in Figure 7.39. This menu is used to generate a graph of a component of the bearing deformation as a function of time as shown in Figure 7.40. The six components of motion that can be graphed are the 3 translation motions u_x, u_y and u_z, and the three rotation components θ_x, θ_y and θ_z of bearing race 1 with respect to bearing race 2. The components are measured in bearing race 2 reference frame. The rotation components are displayed in Radians.

7.20 The BRGREACTION command

The BRGREACTION command of the post-processing menu (Figure 7.3) leads to the menu shown in Figure 7.41. This menu is used to generate a graph of a component of the bearing reaction as a function of time as shown in Figure 7.42. The six force components that can be graphed are the 3 forces F_x, F_y and F_z, and the three moments M_x, M_y and M_z. These components are the forces and moments exerted by race 1 on race 2. The components are calculated in the race 2 reference frame. The moments are about the origin of race 2.

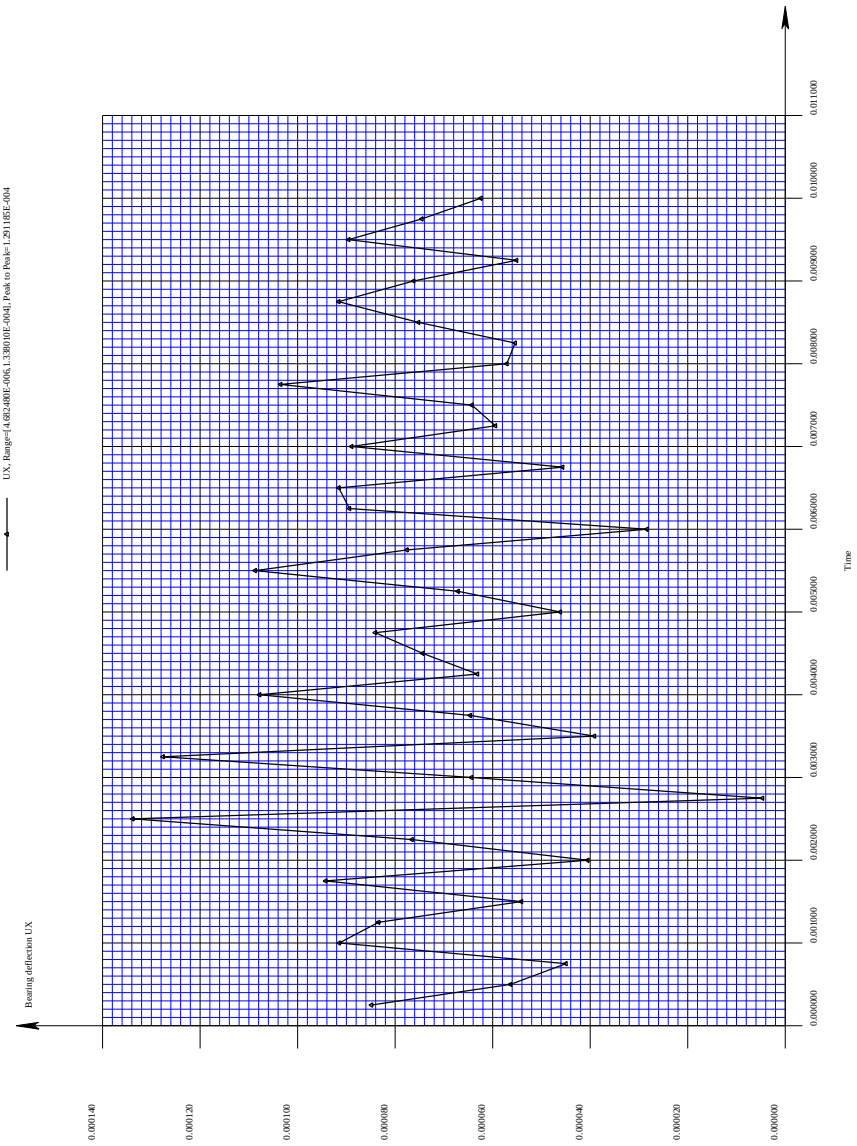


Figure 7.40: The graph generated by the BRGDEFORMN menu.

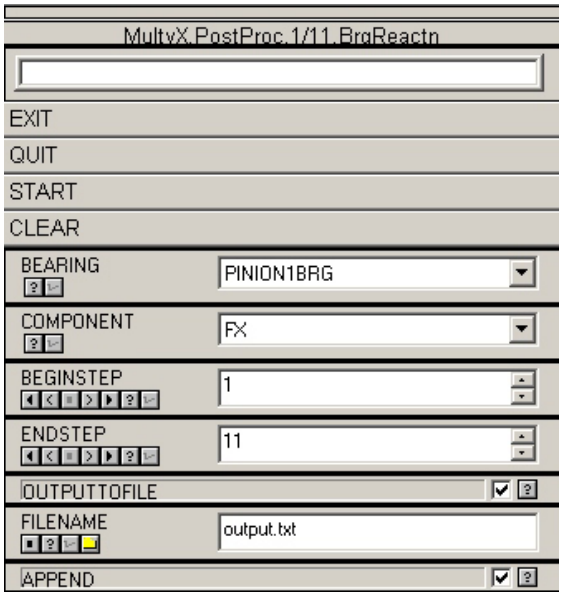


Figure 7.41: The BRGREACTION menu.

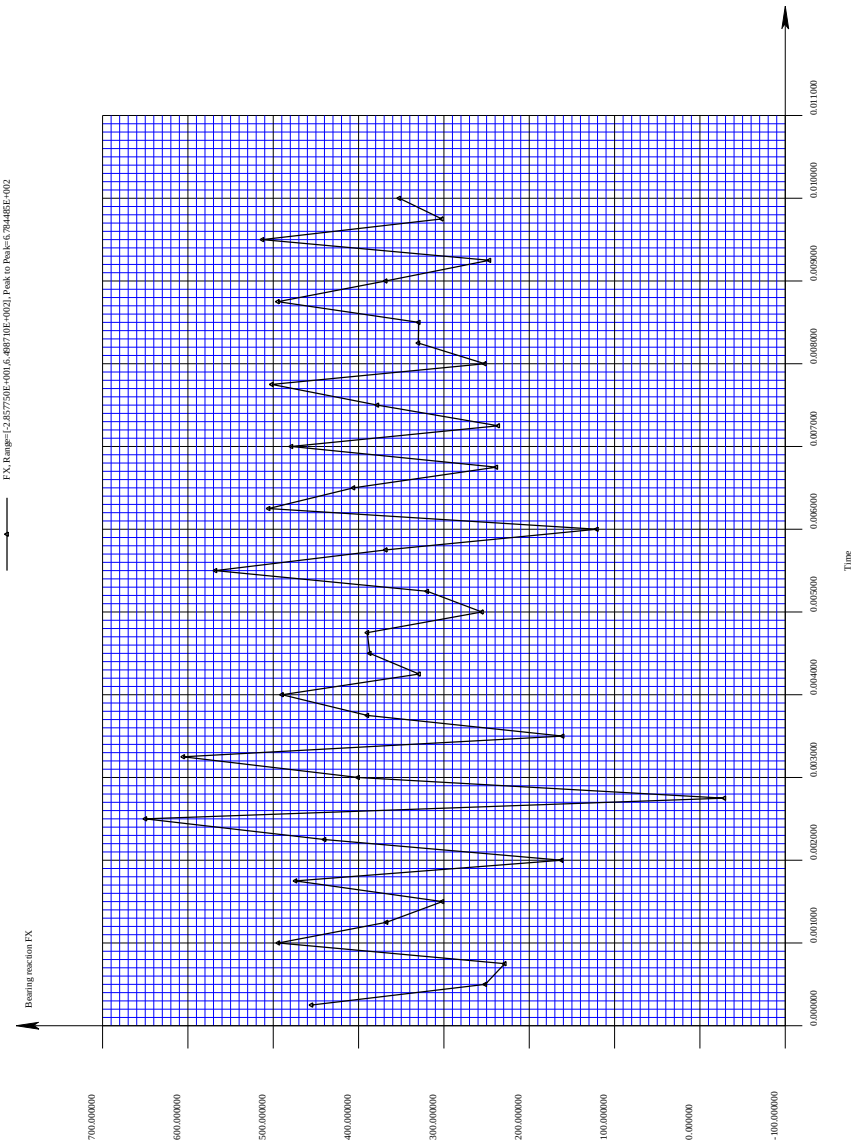


Figure 7.42: The graph generated by the BRGREACTION menu.

Chapter 8

Pre and Post processing using IglassViewer

IglassViewer is a very powerful tool for pre and postprocessing gear models and results. Several features have been added to the Multyx program so as to enhance the compatability with IglassViewer. Thus it can be considered as a program which enables the user to view pre and postprocessing files generated by an external code. Note that the IglassViewer graphics window is independent of the guide graphics window. The advantage of using IglassViewer over guide program for pre and postprocessing is that it is more faster, efficient and more simple to operate. Also, you can visualise the models in their dynamic mode which is not possible using the Guide program. Following sections gives a detailed explanation of the procedure for creating the pre and postprocessing iglass files and also the various functions associated with the iglass program.

8.1 Generating an Iglass file for preprocessing

The GENIGLASSFILE command in Figure 7.1 will lead to a menu shown in Figure 8.1 using which you can generate a preprocessing file for Iglass. The filename is specified in the IGLASSFILENAME menu. The time at which the user wants to visualise the model can be specified in the TIME menu. The user can also visualise the model at a sequence of time steps by entering the number of steps in the NTIMESTEPS menu. The DELTATIME menu is the value of time increment between successive writes to the iglass file. The POPUPIGLASS menu if turned on will automatically open up the Iglass graphical window after the Igass file is generated. If it is not turned on, only the data file for iglass will be created, and iglass will have to be started manually. Using the SELECT menu in Figure 8.1 the user can select the bodies to be displayed in the Iglass graphical window. Click on the START button in Figure 8.1 to generate the Iglass preprocessing file. After the file is generated and if the POPUPIGLASS menu is turned on a separate Iglass window will open showing the reference axes and the gear bodies (selected in the SELECT menu). An example of the Iglass preprocessing window for a planetary system is shown in Figure 8.2. As shown in Figure, it has 3 menus- View, Bodies and Attributes. The Attributes menu is used more commonly in the postprocessing mode. The 'Exit' button in each menu will close the Iglass graphics window.

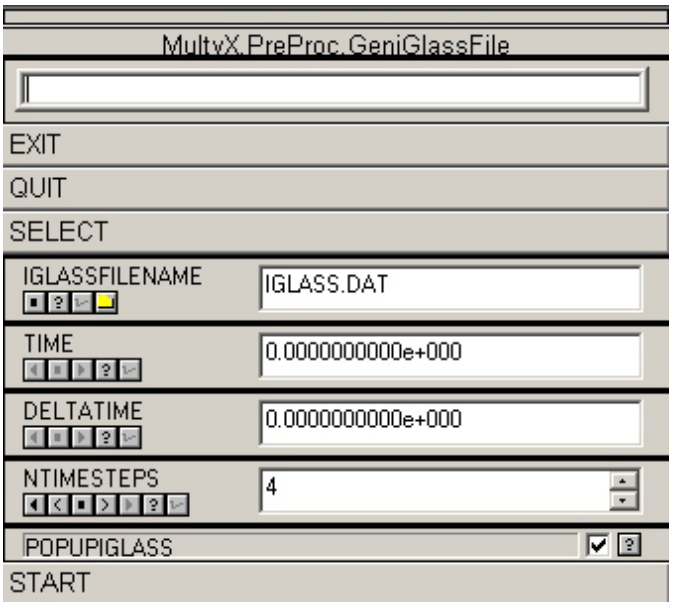


Figure 8.1: The generate Iglass file menu

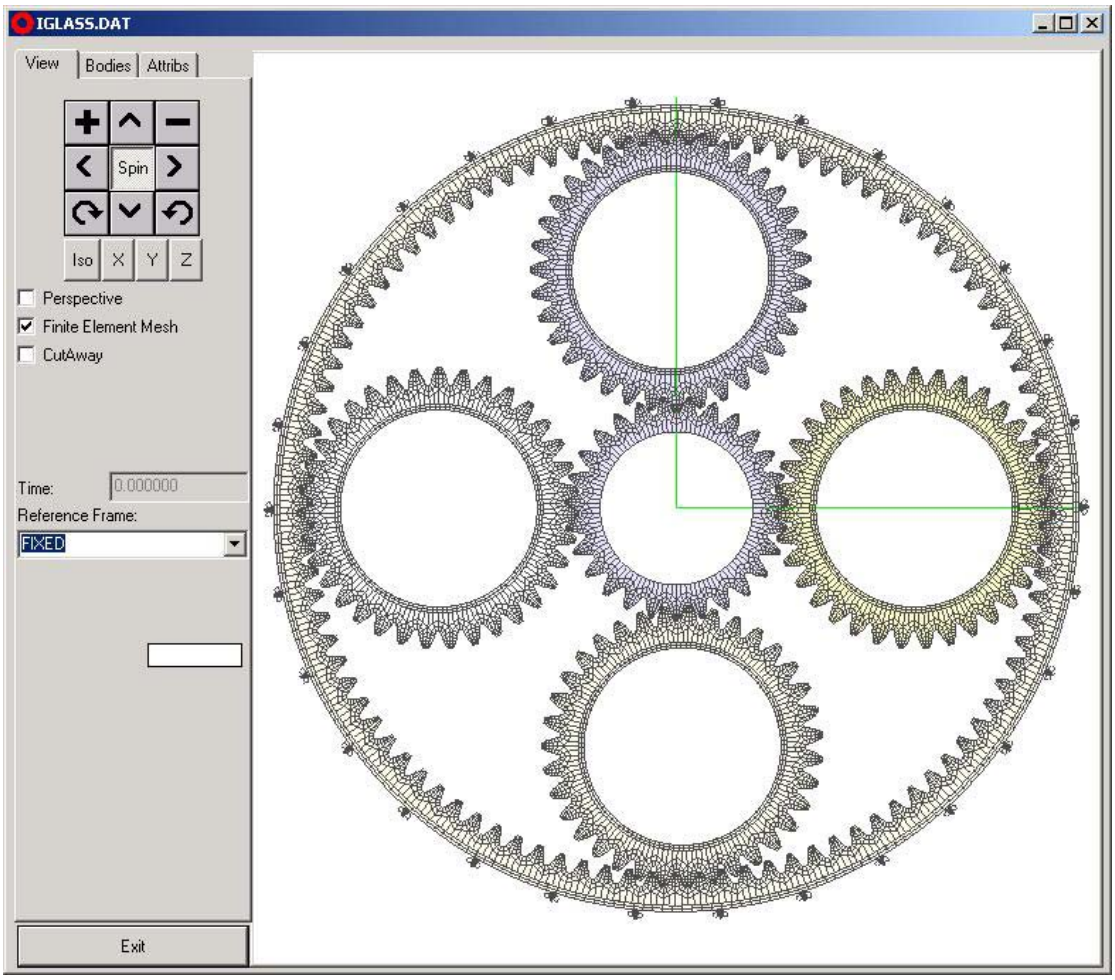


Figure 8.2: An example of an Iglass preprocessing window.

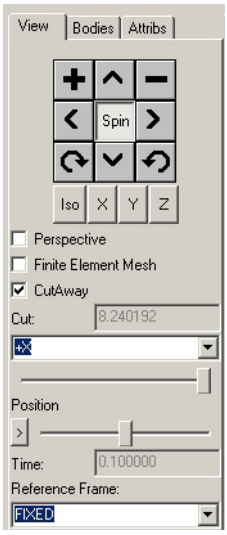


Figure 8.3: Iglass preprocessing view menu

8.2 View menu

The View menu is shown in Figure 8.3. Table 8.1 shows the common tasks performed by some of the buttons displayed in the Iglass window.

Apart from all the features shown in Table 8.1 you can also rotate the model using the left mouse button. Drag the left mouse button in the direction you want to rotate the model in the iglass graphics window. Also the model can be moved in the graphics window in any directions you want using the right mouse button. Drag the right mouse button in the direction you want to move the model in the iglass graphics window.

8.2.1 Finite element mesh

The finite element mesh model can be visualised if the 'Finite Element Mesh' item is selected. Figure 8.4 shows the finite element mesh model of the gear bodies in iglass preprocessing.

8.2.2 Cutting plane

Using the cutting plane switch shown in Figure 8.5 you can visualise the model along a section. This feature is especially useful in pre and post processing of complicated models with a large number of internal gears. The cutting plane can be selected along the +ve and -ve X, Y and Z axes. Using the button below the cutplane switch you can select the cutting plane at various points along the axis chosen by the cut plane switch option.

8.2.3 Selecting the time step

User can visualise the model at a particular timestep in iglass pre-processing using the 'Position' slider shown in Figure 8.6. Each position corresponds to the DELTATIME selected in the generate iglass file menu. The corresponding time can be seen in the 'Time' item shown in Figure 8.7.

Table 8.1: Common buttons in Iglass pre and postprocessing window

Button	Purpose
	Zoom In
	Zoom Out
	Move the model upwards (If Spin is turned OFF)
	Move the model downwards (If Spin is turned OFF)
	Move the model towards right (If Spin is turned OFF)
	Move the model towards left (If Spin is turned OFF)
	Rotate the model upwards (If Spin is turned ON)
	Rotate the model downwards (If Spin is turned ON)
	Rotate the model towards right (If Spin is turned ON)
	Rotate the model towards left (If Spin is turned ON)
	Rotate the model clockwise (If Spin is turned ON)
	Rotate the model counterclockwise (If Spin is turned ON)
	View the model in an isometric view
	View the model in the $Y - Z$ plane
	View the model in the $X - Z$ plane
	View the model in the $X - Y$ plane

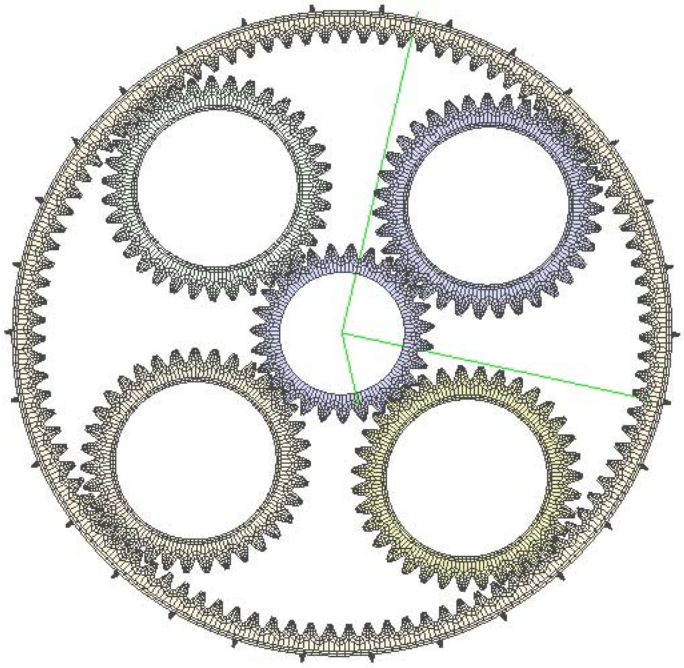


Figure 8.4: Finite element mesh model of the gear bodies

Figure 8.5: The cutting plane switch.

Figure 8.6: The position slider.

Figure 8.7: The time menu.

Figure 8.8: The reference frame switch.

8.2.4 Reference frames

The default reference frame is the FIXED reference frame. All the bodies appear to move when observed from the FIXED frame. The model will align itself to this reference frame when the iglass window pops up. The reference frame can be aligned to a body member using the reference frame switch shown in Figure 8.8. If you select the SUN as the reference frame the reference frame origin will coincide with the origin of the Sun gear. The Sun gear will appear stationary when observed from the SUN reference frame, and rest of the bodies orbits around it. If the PINION option is selected then the reference frame origin aligns itself to the origin of the pinion.

8.3 The Bodies menu

The 'Bodies' menu is shown in Figure 8.9. The body member can be turned on or off by clicking on the member name in the Bodies menu. User can view the tooth and the rim sector separately for each gear body.

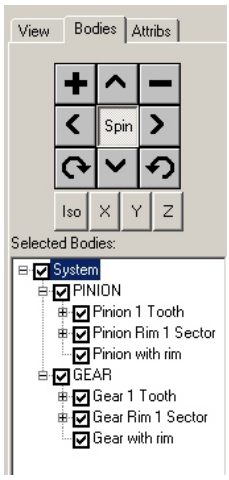


Figure 8.9: Iglass preprocessing Bodies menu

8.4 Post processing using iglass

The GENIGLASSFILE command in Figure 7.3 leads to the generate iglass file menu shown in Figure 8.10 for post processing in iglass. BEGINSTEP and ENDSTEP menus shown in Figure 8.10 define the range for which you want to check for results. Note that these menus are independent of the GOTOPOSN menu shown in Figure 7.3.

An example of an iglass post processing window is shown in Figure 8.11.

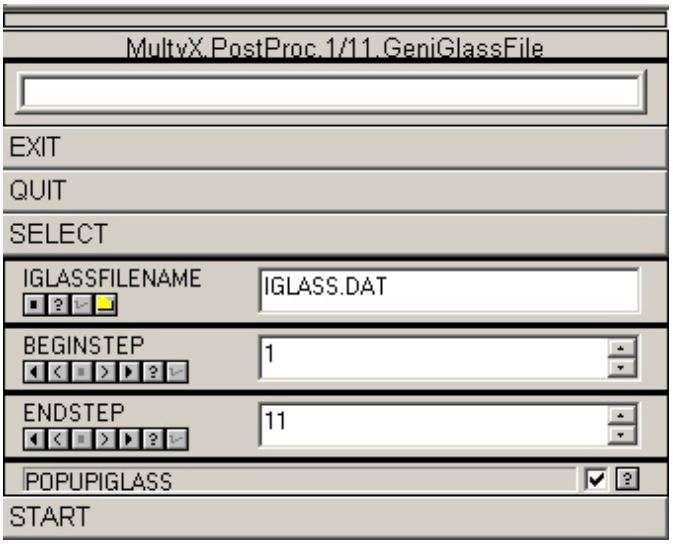


Figure 8.10: The generate iglass file menu for post processing.

8.5 Features specific to iglass post processing

The position switch shown in Figure 8.12 is used to run the simulation of the model in the post processing iglass window. You can look at the simulation at a particular time step by dragging the slider along the scale. The 'Defmn'(deformation) slider shown in Figure 8.13 is used to view the deformed shaped of the gear bodies. The 'Rigid Defl' and the 'F.E.Defl' shows the rigid body deflection and the finite element deflection of the bodies. The magnification scale of deformation can be adjusted using the slider. The load slider shown in Figure 8.14 is used to look for the load patterns on a tooth over the range of time step selected in the BEGINSTEP and ENDSTEP menus. The magnification scale of loading can be adjusted using the slider. The directions of the bearing forces and moments can be visualised using the 'Brg Frc' and 'Brg Mom' sliders shown in Figure 8.15. The magnification scale of the forces and the moments can be adjusted using the respective sliders.

The 'Attribs' menu is shown in Figure 8.16. The attribute menu shown in Figure 8.17 is used to check for contours for different component of results. The available options are DISPLVECTOR, MAXPPLNORMAL, S2PPLNORMAL, MINPPLNORMAL, MAXSHEAR, VONMISES and ERRORESTIMATE. The DISPLVECTOR will pop up a component switch using which the contour for displacement vector in the X, Y and Z directions can be displayed. MAXPPLNORMAL, S2PPLNORMAL, MINPPLNORMAL, MAXSHEAR, VONMISES menus show their respective stress contours. The ERRORESTIMATE menu is used to display the stress error estimate. This error estimate is computed from the magnitude of the inter-element stress discontinuity.

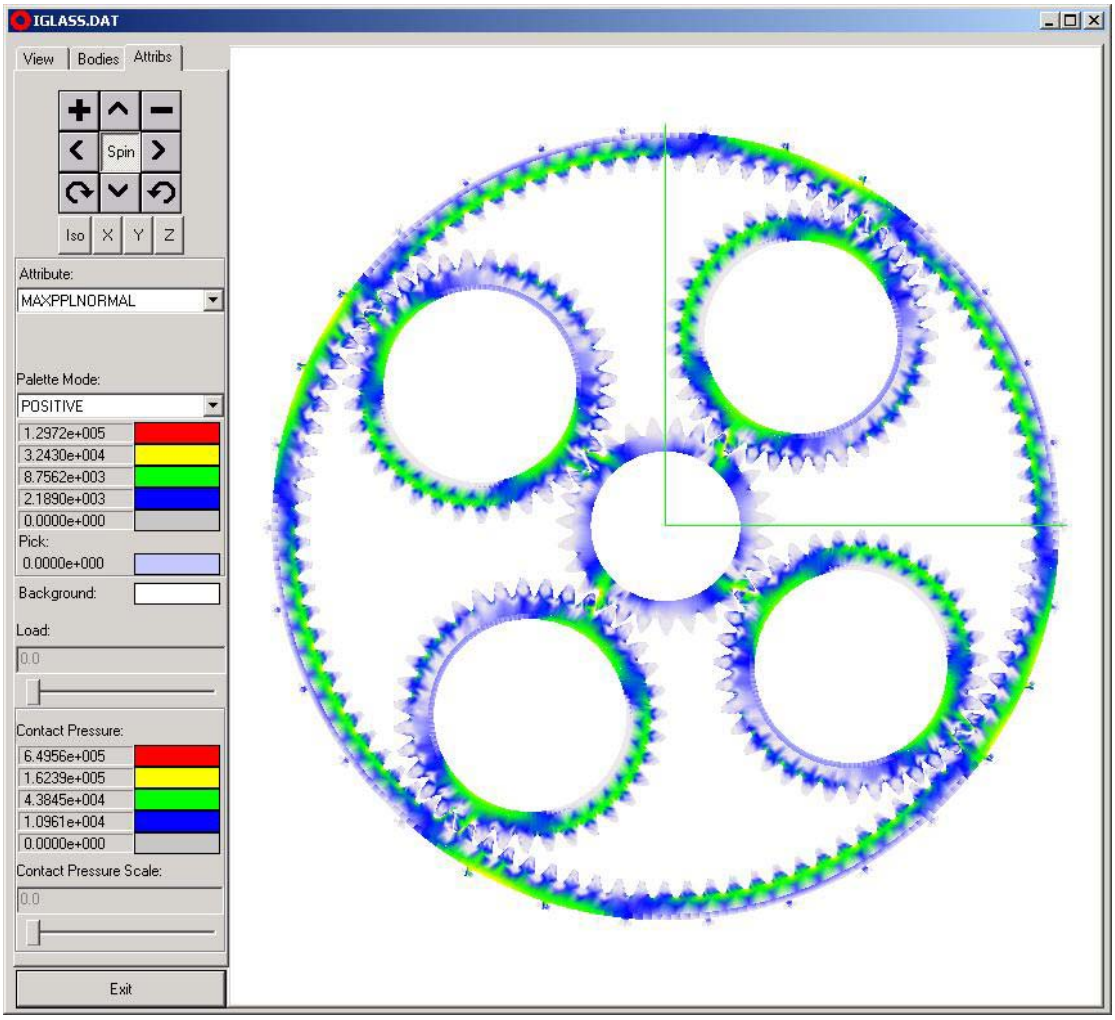


Figure 8.11: An example of an iglass post processing window.

The colors for minimum and maximum stress contours can be controlled using the palette mode menu shown in Figure 8.18. A POSITIVE mode will align the scale from 0 (minimum stress) to a maximum positive value (maximum stress). A NEGATIVE mode will align the scale from 0 to a negative value. The BOTH type mode will align the scale from the maximum negative value (minimum stress) to a maximum positive value (maximum stress). So as to find the stress at a node, double click on the gear body. The finite element nodes are now visible as shown in figure 8.19. Clicking once on the node will show the stress at that nodal point in the 'pick' item of the Palette menu.

Double clicking on the 'Background' button will popup the 'Color' window shown in Figure 8.20 using which you can change the background color of the iglass graphics window.

The Contact pattern menu shown in Figure 8.21 is used to view the contact pressure pattern on the contacting surfaces.

The EXIT button will take you out of the iglass post processing window.

Figure 8.12: The position slider.

Figure 8.13: The deformation slider.

Figure 8.14: The load slider.

Figure 8.15: The bearing forces and moments sliders.

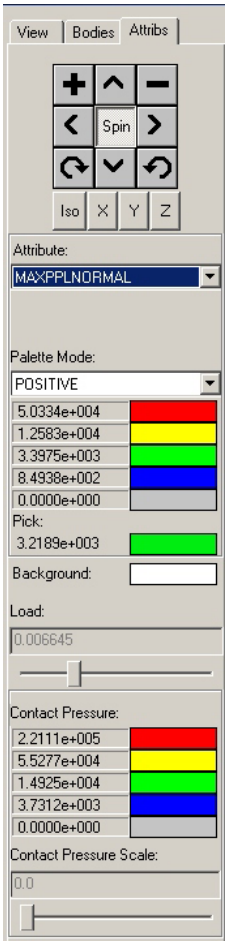


Figure 8.16: The iglass postprocessing attribute menu.

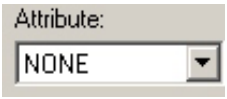


Figure 8.17: The attribute switch.

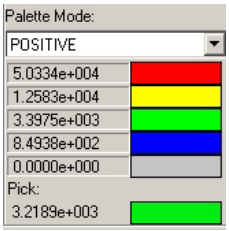


Figure 8.18: The palette switch.

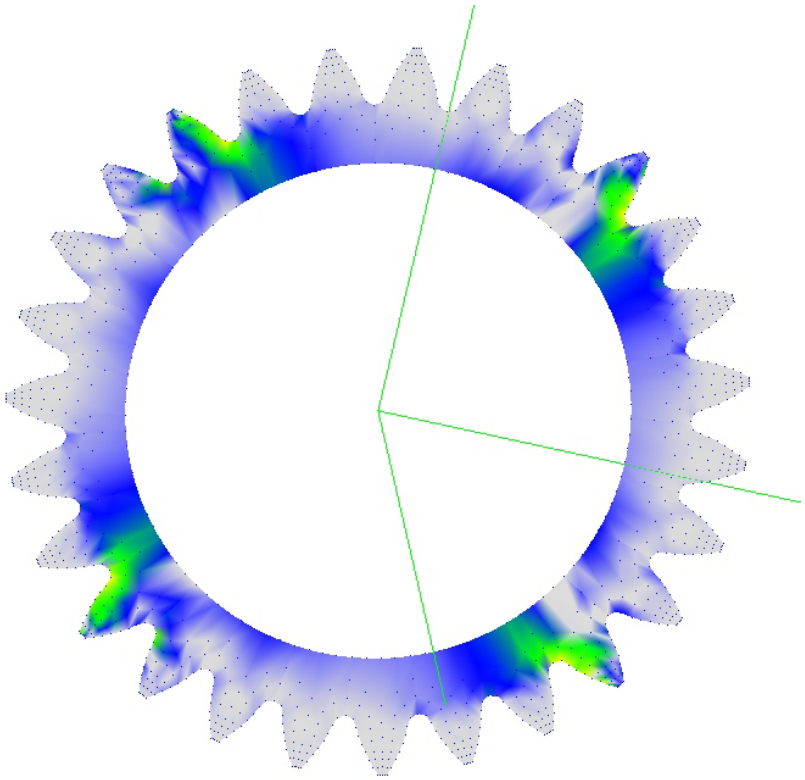


Figure 8.19: Finite element mesh so as to find the stress at a nodal point

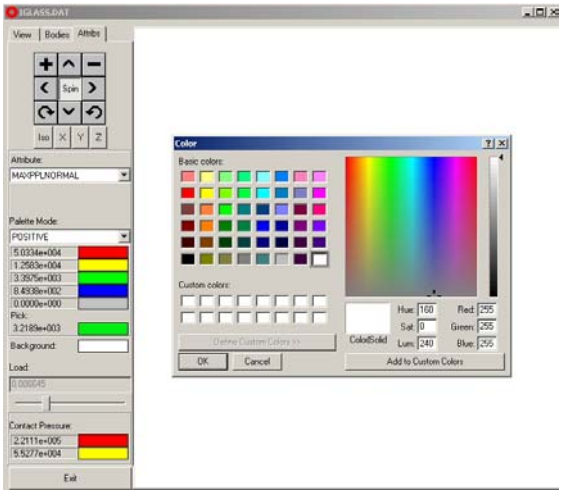


Figure 8.20: The background color popup window switch.

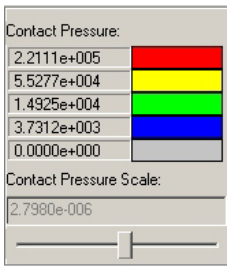


Figure 8.21: The Contact pattern menu.

Chapter 9

Running a dynamic case

Using the *Planetary2D* program for running dynamic cases is one of the most important applications of the *Planetary2D* analysis package. Bearing reactions and Housing reaction plots can be extremely useful in accurately designing the gears for various applications. An example of running a dynamic case is explained in this section. User can initially use this example to get acquainted with running dynamic applications using the *Planetary2D* analysis package.

9.1 The EDIT menu

All the data describing the model is entered in the submenus of the EDIT menu. The user is free to choose any units for force, time and length. Note that all the inputs should then be in units that are consistent with this choice. The data for the example problem is given in English units. The outputs will also appear in English units. Table 9.1 shows the English and SI units for common physical quantities.

Tables 9.2 through 9.14 show the data used for the dynamic example problem. The names of the bearing files and the mesh files given in this example can be changed according to the choice of the user. Note that the working directory must have all the file names provided by the user in the edit menu.

9.2 The System level data

Table 9.2 shows the 2D Planetary system level data. Use the default values for the items not shown in the table.

9.3 The Angular position data

Table 9.3 shows the angular position data for the pinion. PINIONNO item gives the pinion number and the ANGPOSNPINION item gives the nominal angular position of the respective pinions.

9.4 The Pinion data

Select the Rim option for the pinion. The values of LUMPMASS, LUMPMOMINERTIA and LUMPALPHA are all 0.0. Table 9.4 shows the data related to the Pinion tooth. We have not applied any Spacing errors for the pinion.

Table 9.1: Physical units

Physical quantity	English	Metric Engg. units
LENGTH	in	mm
TIME	secs	secs
ANGLE	deg or rad	deg or rad
MASS	lbf.s^2/in	kg
MOMENT OF INERTIA	lbf.s^2.in	kg.mm^2
STIFFNESS	lbf/in	N/mm
SPEED	RPM	RPM
TORQUE	lbf.in	N.mm
YOUNGS MODULUS	lbf/in^2	N/mm^2
DENSITY	lbf.s^2/in^4	kg/mm^3
LOAD	lbf	N
STRESSES	psi	N/mm^2

Table 9.2: System level data

Item	Description
NUMPINIONS	4
CENTERDIST	3.5
INPUT	SUN
TORQUEINPUT	1.62e4
RPMINPUT	-1.623e3
RPMRING	0.0

Table 9.3: Pinion angular position data

Item	Description
NUMPINIONS	4
PINIONNO	1
ANGPOSNPINION	0.0
PINIONNO	2
ANGPOSNPINION	91.42
PINIONNO	3
ANGPOSNPINION	180
PINIONNO	4
ANGPOSNPINION	271.42

Table 9.4: Data describing the tooth profile for pinion

Item	Description
NTEETH	35
DIAMPITCH	8.85
PRESSANGLE	24.6
THICKNESS	0.163
HOBTIPRAD	0.018
OUTERDIA	4.13
ROOTDIA	3.60
INNERDIA	2.90
YOUNGSMOD	3.004e7
POISSON	0.3
DENSITY	7.112e-4
ALPHA	4.79e2
BETA	1.4e-7
MESHFILE	pinion.msh
TEMPLATE	MEDIUM.TPL

Table 9.5: Quadratic tip modification data

Item	Description
ROLLQUADTIPMOD	12.
MAGQUADTIPMOD	1.00e-4.

9.5 The Pinion tooth modification

We apply the Quadratic Tip Modification on the Pinion tooth for this example problem. The modification values are given in the Table 9.5.

9.6 The Pinion bearing data

The RIGIDRACE box for the Pinion bearing is not checked, that means the inner diameter of the Pinion is not rigid. Table 9.6 gives the details of the Pinion bearing data menu.

9.7 The Pinion Rim data

Table 9.7 shows the data for the Pinion rim.

Table 9.6: Bearing menu parameters

Item	Description
CIRCORDER	4
BRGFILE	pinion.brg

Table 9.7: Pinion Rim data

Item	Description
MESHFILENAME	pinionrim.msh
RIMDIA	3.0668
NRADIAL	2
NCIRCULAR	128
ELEMTYPE	LINEAR
CIRCORDER	4

Table 9.8: Data describing the tooth profile for sun gear

Item	Description
NTEETH	27
DIAMPITCH	8.85
PRESSANGLE	24.6
THICKNESS	0.176
HOBTIPRAD	0.04256
OUTERDIA	3.309
ROOTDIA	2.7775
INNERDIA	2.25
YOUNGSMOD	3.004e7
POISSON	0.3
DENSITY	7.112e-4
ALPHA	4.79e2
BETA	1.4e-7
MESHFILE	sun.msh
TEMPLATE	MEDIUM.TPL

9.8 The Sun gear data

Do not select the rim option for the Sun gear. The values of LUMPMASS, LUMPMOMINERTIA and LUMPALPHA are all 0.0 for this example. Also, we do not apply the spacing errors for the Sun gear in this problem. The inner diameter of the Sun is assumed to behave like a rigid cylinder. Also, the Sun gear has a bearing. Table 9.8 shows the Sun gear tooth data.

9.9 The Sun tooth modification

We apply the tabular profile modification on the sun gear tooth for this example problem. The modification values are given in Table 9.9.

9.10 The Ring gear data

Select the SPLINEOPTION (splines will be modeled) and the RIMOPTION (separate finite element model will be generated between the Rim dia of the Ring gear and the Outer dia). Select the default value (32) for the CIRCORDER item. We do not apply any Spacing errors for the ring gear tooth in this example problem. Table 9.10 shows the ring gear tooth data.

Table 9.9: Tabular profile modification data for the Sun gear tooth

Item	Description
NROLLS	4
ROLLNO	1
ROLLANGLE	18.26
MAGNITUDE	0.00
ROLLNO	2
ROLLANGLE	31.85
MAGNITUDE	-1.5e-4
ROLLNO	3
ROLLANGLE	36.42
MAGNITUDE	1e-4
ROLLNO	4
ROLLANGLE	37.37
MAGNITUDE	5e-4

Table 9.10: Data describing the tooth profile for Ring gear

Item	Description
NTEETH	99
DIAMPITCH	9.1428
PRESSANGLE	20.19
THICKNESS	0.123
FILLETRAD	5.8e-2
MINORDIA	10.69
ROOTDIA	11.18
OUTERDIA	11.85
YOUNGSMOD	3.004e7
POISSON	0.3
DENSITY	7.112e-4
ALPHA	4.79e2
BETA	1.4e-7
MESHFILE	ring.msh
TEMPLATE	MEDIUM.TPL

Table 9.11: Tabular profile modification data for the Ring gear tooth

Item	Description
NROLLS	3
ROLLNO	1
ROLLANGLE	20.63
MAGNITUDE	0.00
ROLLNO	2
ROLLANGLE	19.16
MAGNITUDE	8e-4
ROLLNO	3
ROLLANGLE	18.84
MAGNITUDE	1.05e-3

Table 9.12: Spline data for the Ring gear

Item	Description
CONTACTTYPE	DOUBLESIDED
ANGULARPLAY	7.4e-2
NSPLINES	50
ANGLEFIRST	0.0
ELEMTYPE	LINEAR
NWIDTH	4
NHEIGHT	4
WIDTH	0.1
HEIGHT	0.1323
PRESSANGLE	14.5

9.11 The Ring gear tooth modification

The tabular profile modification is applied on the ring tooth for this problem. The values are given in the Table 9.11.

9.12 The Spline data for the Ring gear

For this example problem we have evenly spaced the splines over the circumference of the Ring gear. Table 9.12 shows the spline data for the ring gear.

9.13 The Ring gear rim data

Table 9.13 shows the Ring gear rim data.

9.14 The Carrier data

The carrier for this problem is modeled as a rigid body and it has a bearing. Bearing file name is provided in the BRGFILE box. The data related to the carrier menu is shown in the Table 9.14.

Table 9.13: Rim data parameters for the ring gear

Item	Description
MESHFILENAME	ringrim.msh
RIMDIA	11.675
NRADIAL	2
NCIRCULAR	256
CIRCORDER	32

Table 9.14: Carrier data

Item	Description
BRGFILE	carrier.brg
LUMPMASS	100
LUMPMOMINERTIA	1000
LUMPALPHA	0.0

9.15 Calculating the time step for dynamic problems

Let,

$$
\begin{aligned}
\omega_s &= \text{Angular speed of the Sun gear in rad/sec} \\
\omega_c &= \text{Angular speed of the Carrier in rad/sec} \\
\omega_p &= \text{Angular speed of the Pinion gear in rad/sec} \\
\omega_R &= \text{Angular speed of the Ring gear in rad/sec} \\
RPM_s &= \text{Speed of the sun gear in RPM} \\
RPM_c &= \text{Speed of the carrier in RPM} \\
RPM_p &= \text{Speed of the pinion in RPM} \\
RPM_R &= \text{Speed of the ring gear in RPM} \\
N_s &= \text{Number of teeth on the sun gear} \\
N_p &= \text{Number of teeth on the pinion gear} \\
N_R &= \text{Number of teeth on the ring gear} \\
\Delta_T &= \text{Time Step} \\
T_{cycle} &= \text{Tooth Cycle time} \\
n_{steps} &= \text{Total number of steps per tooth cycle}
\end{aligned}
$$

The following relationships are used to determine the time step required for dynamic cases.

$$(\omega_s - \omega_c)N_s = -(\omega_p - \omega_c)N_p = -(\omega_R - \omega_c)N_R$$

$$T_{cycle} = |\frac{2\pi}{(\omega_s - \omega_c)N_s}| = |\frac{2\pi}{(\omega_p - \omega_c)N_p}| = |\frac{2\pi}{(\omega_R - \omega_c)N_R}|$$

$$T_{cycle} = |\frac{60}{(RPM_s - RPM_c)N_s}| = |\frac{60}{(RPM_p - RPM_c)N_p}| = |\frac{60}{(RPM_R - RPM_c)N_R}|$$

$$\Delta_T = \frac{T_{cycle}}{n_{steps}}$$

9.16 The setup menu

For the example problem we assume that the system is starting from an undeformed state. The analysis is divided in to 3 ranges.The purpose of using 3 ranges is to minimize initial transients.

The first range is the Static range when the input speed is 0.00. The second range is dynamic when the input speed increases from 0 r.p.m to 1623 r.p.m. The ramping from minimum speed to maximum speed is controlled by the STARTSPEEDFACTOR parameter. The STARTSPEEDFACTOR for range 1 and range 2 is 0.0. The third step is again dynamic when the input member (Sun gear for this example) maintains a constant speed of $1623rpm$. The STARTSPEEDFACTOR for range 3 is 1.00. The solution method for the Static range is STATIC and that for the dynamic ranges we use the NEWMARK method.

For this problem we run the analysis for about 3000 steps (N_{steps} for range 3 = 3000). For so many steps normally the postprocessing file size goes beyond 2 Gigabytes which may be too large. So generally we do not write the postprocessing file while running dynamic cases (unless it is for less than 100 steps). So do not check the POSTPROCWRITE box. Also most of the results are extracted using Surface gages, FE probes and bearing files. All the other parameters related to the setup menu are given in Table 9.15.

For the dynamic example problem,

$RPM_s = 1623$ r.p.m.

$RPM_R = 0$ r.p.m.

$N_s = 27$.

$N_R = 99$.

N_{steps} for first (Static) range = 1.00

N_{steps} for second (Dynamic) range = 1000

N_{steps} for third (Dynamic) range = 3000

Using the above data and the formulas,

T_{cycle} = 1.742E-3 secs.

Time step (For 100 steps per tooth cycle) = 1.742E-5 secs.

Tooth meshing frequency = $1/T_{cycle}$ = 574.05 Hz.

Table 9.15: Setup menu

Item	Description
SEPTOL	3e-3
NPROFDIVS	3
DSPROFSUN	0.2
DSPROFRING	0.2
INITIALTIME	0.0
NRANGES	3
RANGE	1
SOLMETHOD	STATIC
NTIMESTEPS	1
DELTATIME	1.742E-5
STARTSPEEDFACTOR	0.0
STARTTORQUEFACTOR	1.0
ENDTORQUEFACTOR	1.0
RANGE	2
SOLMETHOD	NEWMARK
NTIMESTEPS	1000
DELTATIME	1.742E-5
STARTSPEEDFACTOR	0.0
STARTTORQUEFACTOR	1.0
ENDTORQUEFACTOR	1.0
RANGE	3
SOLMETHOD	NEWMARK
NTIMESTEPS	3000
DELTATIME	1.742E-5
STARTSPEEDFACTOR	1.0
STARTTORQUEFACTOR	1.0
ENDTORQUEFACTOR	1.0
NSTEPSAVE	500
SAVEFILENAME	restper.dat
OUTPUTFILENAME	restout.dat

9.17 Analysis and Results

Before starting an analysis, sensor locations have to be set up to measure stresses and loads in the model. User should place the surface gages, feprobes and load sensors at appropriate locations depending upon the requirement of results. Once the sensors are set up click on the generate model button to check for any errors or warnings. If there are none then click on the STARTANAL button to start the analysis.

The *Planetary2D* analysis package creates files like PINIONnBRGRES.DAT, SUNBRGRES.DAT, HOUSINGRES.DAT, GAGES.DAT, FEPROBES.DAT, LOADS.DAT etc after the analysis is complete. All these files are created in the working directory. Figure 9.1 shows the plot of maximum principal normal stress for the fillet surface on the sun gear using the data from GAGES.DAT. The information for the Surface gage whose output is shown in Figure 9.1 is described in Table 9.16. Each row in the file GAGES.DAT corresponds to a time instant. The first column in the file contains the value of time. The remaining columns contain the data for the surface gages. There are 4 columns of data for each gage. So as to obtain the plot shown in the figure we have used the data from the first(critical maximum principal normal stress) and the second(critical minimum principal normal stresses) column for Gage no.1.

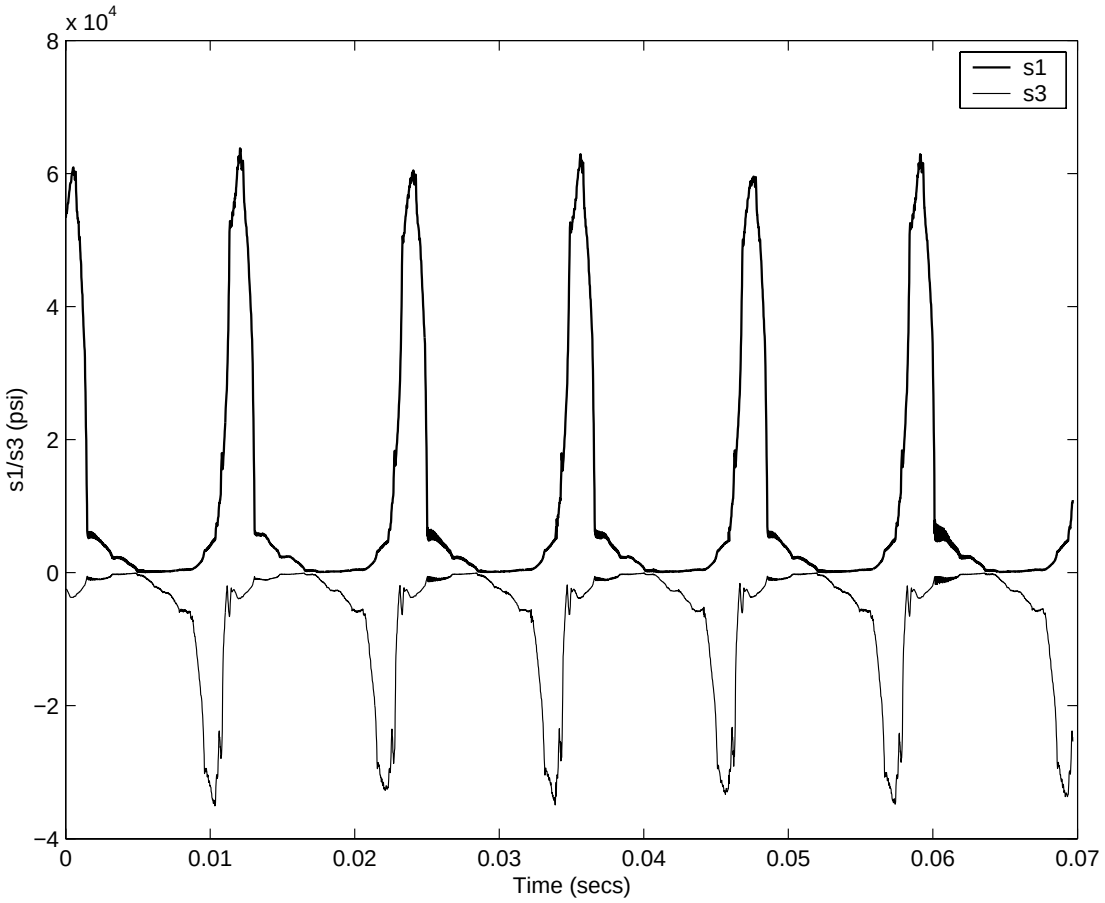


Figure 9.1: Graph showing the dynamic maximum principal normal stresses (s1) and minimum principal normal stresses (s3) against time for the fillet surface of the sun gear, measured by Surfage Gage 1. The results are obtained from the file GAGES.DAT.

Table 9.16: Surface gage data for extracting the plot shown in Figure 9.1

Item	Gage 1
BODY	SUN
SURFACE	FILLET1
TOOTHBEGIN	1
TOOTHEND	1
SPROFBEGIN	0.00
SPROFEND	15.00
NUMSPROF	51
TFACEBEGIN	0.00
TFACEEND	0.00
NUMTFACE	1
DEPTHBEGIN	0.00
DEPTHEND	0.00
NUMDEPTH	1
DISTMIN	0.005
FILENAME	GAGES.DAT

The dynamic analysis should attain a steady state behavior after some time. This can be checked for using the pinion bearing reaction plots as shown in the Figure 9.2. This plot shows the pinion bearing tangential displacement against time. The data for the plot is obtained from the PINION1BRGRES.DAT file. The file contains the results for Pinion1 Bearing. It has one row for each instant of time analyzed. The first column contains the time. The third column will give the tangential displacement for Pinion1. Reaction forces transmitted in the Housing can be checked using the housing bearing reaction plots. A plot of timestep against housing moments is shown in Figure 9.3. Column no.13 in the HOUSINGRES.DAT file gives the moment about the Z-axis for each time instant.

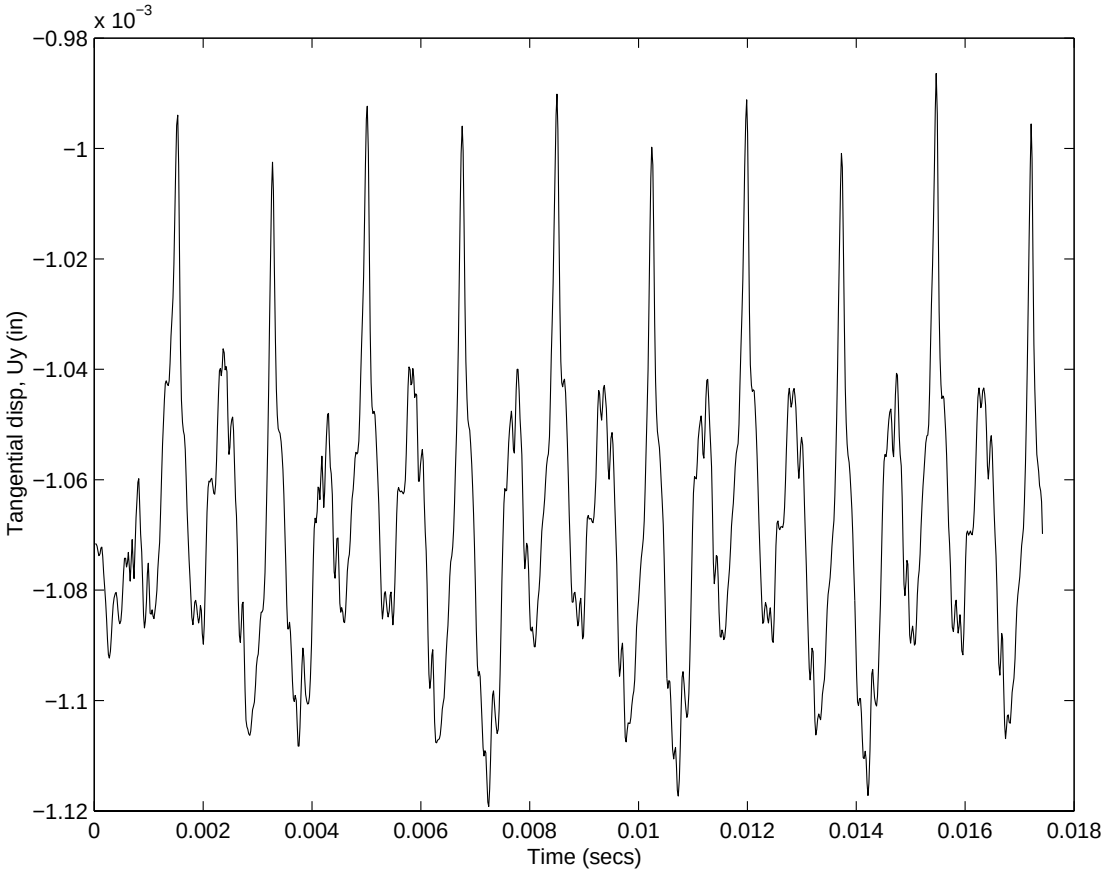


Figure 9.2: Graph showing the pinion bearing tangential displacement against time for dynamic analysis. The tangential displacement at each instant of time is given in Column no.3 of the PINION1BRGRES.DAT file.

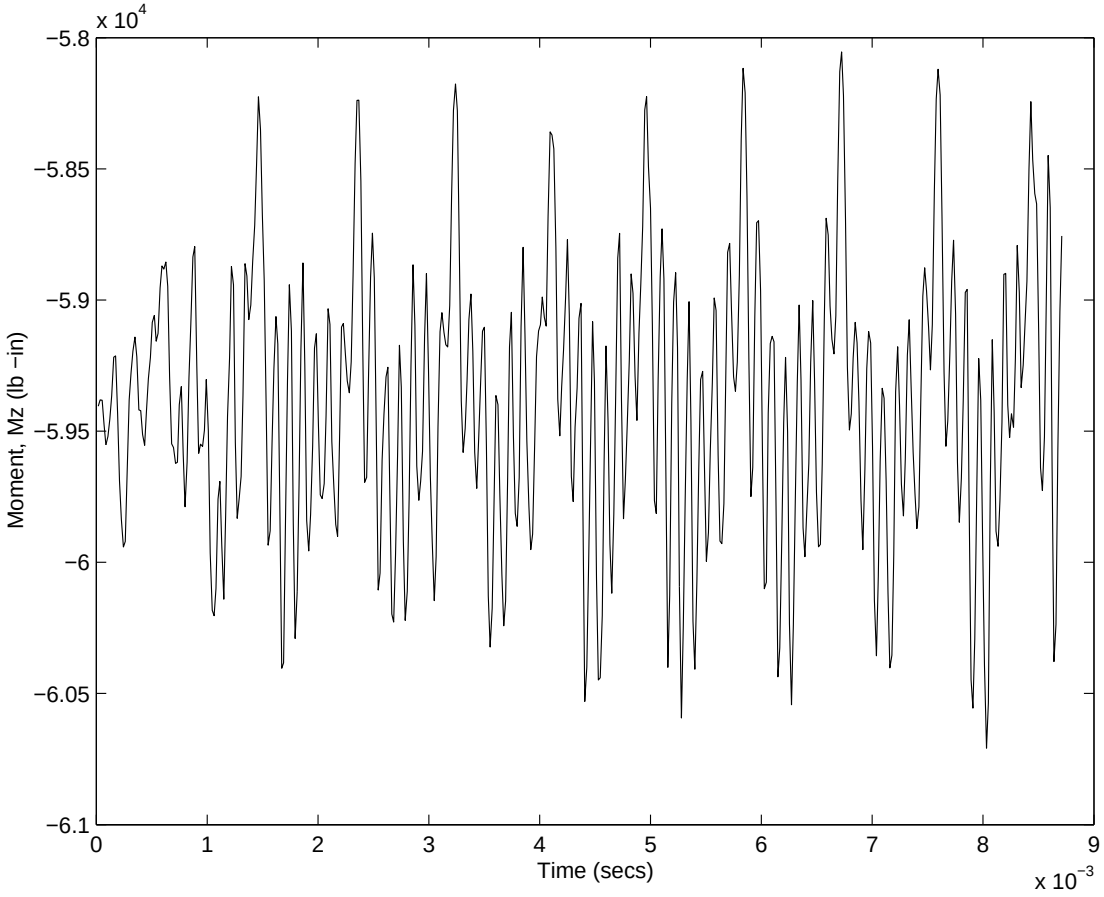


Figure 9.3: Graph showing the housing moments against time for dynamic analysis. The moment about the Z-axis for each time instant is given in column no.13 of the HOUSINGRES.DAT file

9.17.1 Calculating the Dynamic load factor

A graph of timestep against dynamic loads obtained from the file LOADS.DAT is shown in Figure 9.4. The LOADS.DAT file has one row for each instant of time. The first column contains the time. Each subsequent column contains the reading for one load sensor. Table 9.17 shows the data for a load sensor so as to obtain the plot shown in Figure 9.4. So as to obtain the plot of static loads against time a STATIC analysis is performed on the same planetary gear set. Figure 9.5 shows a plot of static load against time using the TOOTHLOAD command. The TOOTHLOAD data used to extract this plot is given in Table 9.18. Using the static and dynamic load values you can calculate the dynamic load factor, a term very commonly used in the gear industry to predict dynamic stresses from static values.

$$\text{Dynamic Load Factor} \quad = \quad \frac{\text{Dynamic Height}}{\text{Static Height}} \tag{9.1}$$

For the present example, the `Dynamic Load Factor` $= \frac{3514.50}{2909.23} = 1.208$

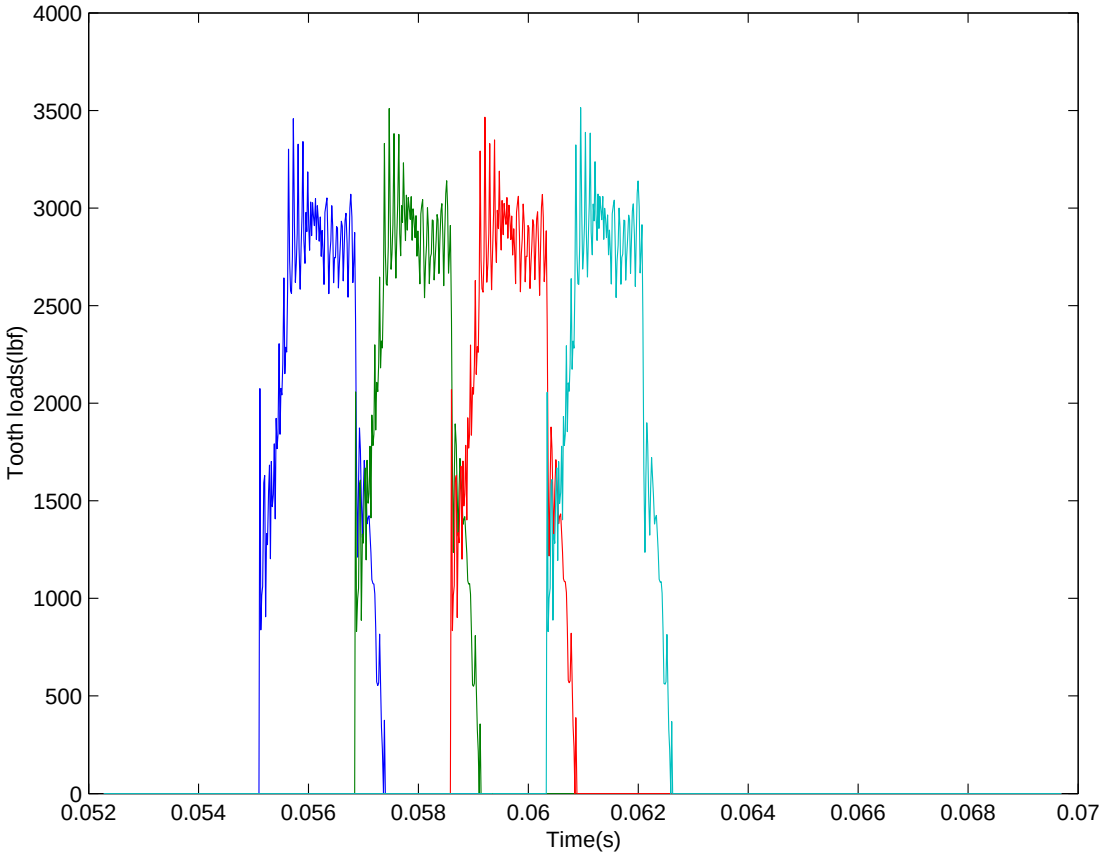


Figure 9.4: Graph showing the dynamic tooth load against time. The plot is obtained using the load values from column nos.6, 10, 11 and 12 from the LOADS.DAT file.

Table 9.17: Load sensor data for extracting the plot shown in Figure 9.4

Item	Description
NLOADSENSORS	11
LOADSENSOR	5
SURFPAIR	PINION1_SURFACE1_SUN_SURFACE1
MEMBER	SUN
TOOTH	1
FILENAME	LOADS.DAT

Table 9.18: Tooth load data for extracting the plot shown in Figure 9.5

Item	Description
SURFACEPAIR	PINION1_SURFACE1_SUN_SURFACE1
MEMBER	SUN
TOOTHBEGIN	1
TOOTHEND	1
BEGINSTEP	1
ENDSTEP	11

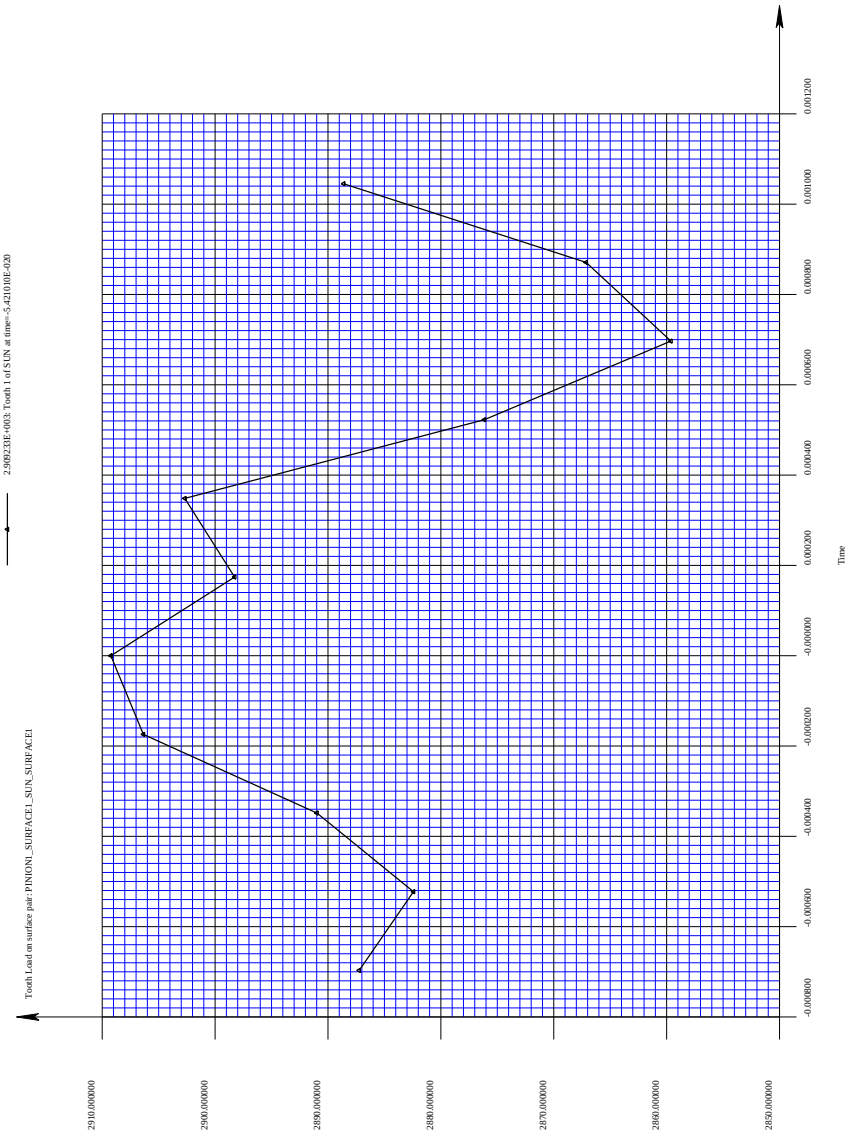


Figure 9.5: Graph showing the static tooth load against time using the TOOTHLOAD command.

9.18 Applying the 2D Planetary program for modal analysis

Following is the detailed procedure which demonstrates the application of the *Planetary2D* program for calculating the modes for the pinion.

- Use the `PINIONnBRGRES.DAT` files for data analysis related to the pinions. The U_x component (column no.2) gives the radial displacement and the U_y component (column no.3) gives the tangential displacement.
- You can either use excel or MATLAB to calculate the fourier transform (fft) of U_x or U_y values. This gives us the frequency response of the pinion. If $X_i(t) = X(t_0 + (i-1)\Delta t)$, $i = 1 : N$, is the time domain response of a particular component of a system, then the fft of X will yield $X_i = X(f_i) = X((i-1)\Delta f)$ where $\Delta f = 1/(N.\Delta t)$. Half of these points lie below the Nyquist frequency $N\Delta f/2$ and half lie above. The frequency response data f_i for $i > N/2$ are usually discarded. The first point X_1 is proportional to the mean response $X_1 = \sum X_i/N$.
- The data obtained for the last range of the dynamic analysis (when the speed is constant) is normally used to plot the frequency response. So for the present example problem $N = 3000$ since N_{steps} for the last step is 3000.
- An example of a MATLAB program for plotting the frequency response (Figure 9.6 & Figure 9.9) and calculating the critical frequency value for Pinion1 is shown below:

```
%Time Step
Dt=1.742e-5;

%No. of steps
N = 3000;

cd C:Planetary2D
load PINION1BRGRES.DAT

x1=PINION1BRGRES(1001:4000,1);
x2=PINION1BRGRES(1001:4000,2);

Df=1/(N*Dt);
for i=1:N,
f(i)=(i-1)*Df;
end;
%Obtain X(f)=FFT of known signal x(t)
%Fourier transform of $U_x$
X=fft(x2,N)*2/N;
%Plotting the frequency response
plot(f(1:N/2),abs(X(1:N/2)));
plot(f(2:N/2),abs(X(2:N/2)));
%Zooming in to catch the peak response
plot(abs(X(2:1000)))
plot(abs(X(2:500)))
plot(abs(X(25:35)))

%frequency
f(31)
ans = 574.0528

%Peak Response
X(31)
ans = 7.6426e-007 -3.2792e-006i

%Absolute value of the peak response
abs(X(31))
ans = 3.3671e-006

%Real part of the peak response
```

```
real(X(31))
ans = 7.6426e-007

%Imaginary part of the peak response
imag(X(31))
ans = -3.2792e-006

%Phase angle
angle(X(31))
ans = -1.3418
```

The frequency distribution using the absolute values can be plotted using the `abs` command as shown in the program. You can plot the frequency response using the real and imaginary values by using the commands `real` and `imag` respectively. The phase angle ϕ can be calculated using the `angle` command.

- By narrowing down the frequency plot (Figure 9.7 & 9.10) you can measure the frequency at which the amplitude the maximum. For this example problem critical frequencies came out to $574.05Hz$, $1148.10Hz$ and $4018.4Hz$ (Figure 9.8 & 9.11) Critical frequency in this case is either the tooth meshing frequency or a multiple of tooth meshing frequency.

- Table 9.19 and 9.20 shows the data obtained from the frequency response plot for the radial and the tangential displacement components for all the pinions. Using this data the absolute values in radial direction (U_x) and tangential direction (U_y) for Pinion1 are $2.9142e-06$ and $6.282e-06$ respectively.

- The equation defining the relationship between the displacement and frequency is given by $U_x = \text{Acos}(2\pi\text{ft} + \phi)$, where A=absolute value of radial displacement and ϕ is the phase angle(radians) in the radial direction. For different values of t you will get a set of values for U_x.

  Similarly tangential component U_y can be calculated using the data from Table 9.20. A graph of U_x Vs U_y for Pinion1 is shown in Figure 9.12. Using this graph you can predict the motion of Pinion1 in the Planetary system in the dynamic state. A graph of U_x Vs U_y for the sun gear is shown in Figure 9.13. Using this graph you can predict the motion of the sun gear in the Planetary system in the dynamic state.

  An example of a MATLAB program for plotting a graph of U_x Vs U_y for Pinion1 is shown below:

```
%Time Step
Dt=pi/2;

%No. of steps
N=30;

%Build arrays containing t and X(t)
for i=0:N-1,
t(i+1)=Dt*i;
end;

%Radial displacement
Ux=3.367e-6*cos(2*pi*574.05*t - 1.3418);
%Tangential displacement
Uy=6.282e-6*cos(2*pi*574.05*t + 1.8201);
plot(Ux,Uy)
```

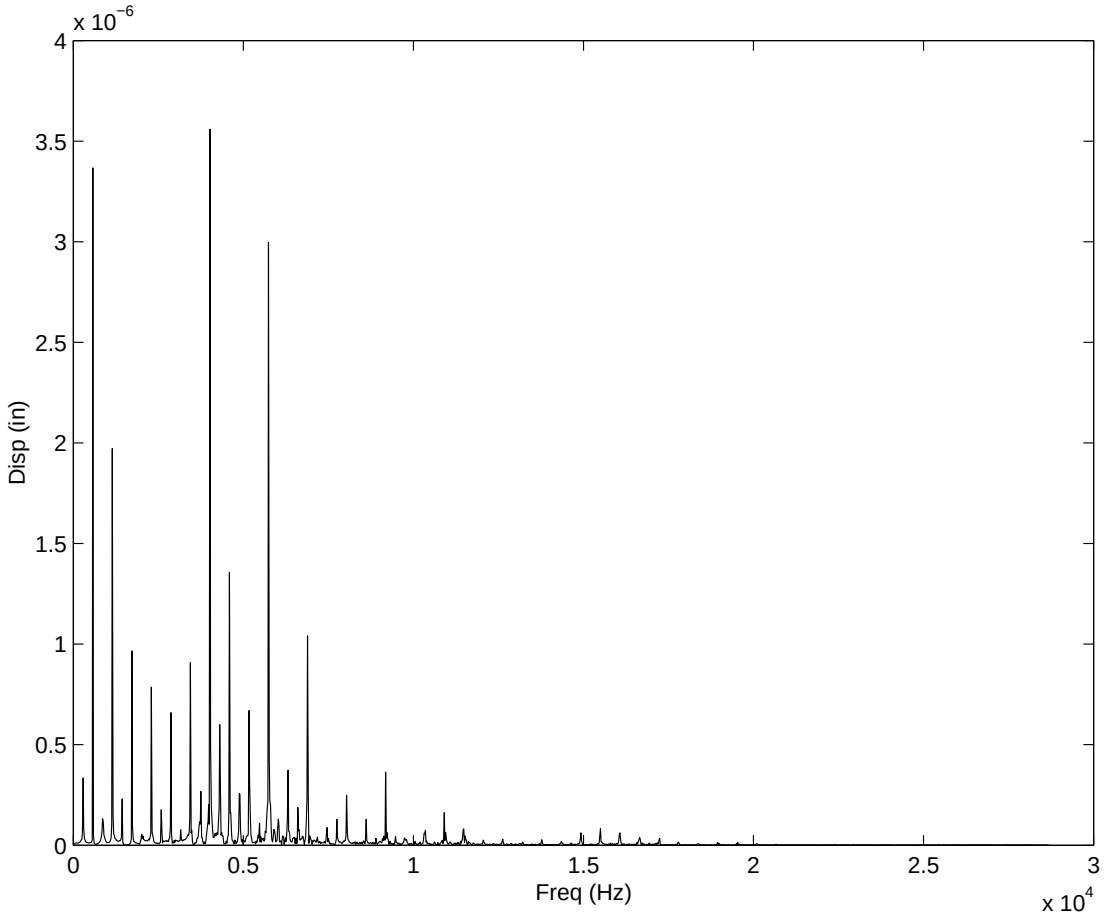


Figure 9.6: Plot showing the radial component frequency response of pinion1 in the planetary system in the dynamic state (transient response is in column no.2 of the PINION1BRGRES.DAT file)

Table 9.19: Displacement values in the radial direction for a critical frequency of 574.05 Hz for the pinions

Body member	Real	Imag	Absolute	Phase
Pinion1	7.6426e-7	-3.2792e-6	3.367e-6	-1.3418
Pinion2	-2.5407e-6	-2.5428e-6	3.5946e-6	-2.3558
Pinion3	-5.0789e-7	3.139e-6	3.1798e-6	1.7312
Pinion4	2.2697e-6	2.6768e-6	3.5095e-6	0.8675
Sun	1.2569e-6	-2.6292e-6	2.9142e-6	-1.1249

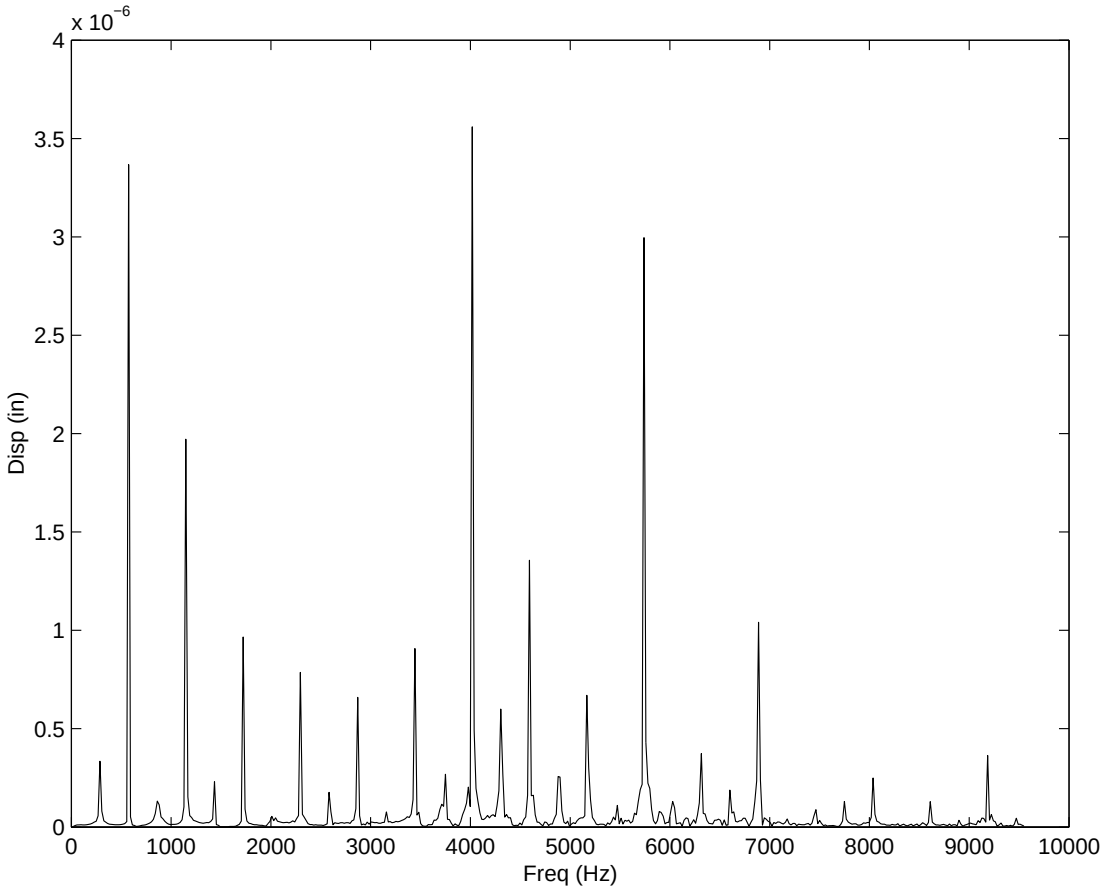


Figure 9.7: Plot showing a narrow range of radial component frequency response of pinion1 in the planetary system in the dynamic state (transient response is in column no.2 of the PINION1BRGRES.DAT file)

Table 9.20: Displacement values in the tangential direction for a critical frequency of 574.05 Hz for the pinions

Body member	Real	Imag	Absolute	Phase
Pinion1	-1.550e-6	6.0878e-6	6.282e-6	1.8201
Pinion2	8.8757e-6	4.1427e-6	9.7949e-6	0.4367
Pinion3	-3.8233e-6	-3.0933e-6	4.9179e-6	-2.4613
Pinion4	-3.4846e-6	-7.1558e-6	7.9591e-6	-2.0240
Sun	-4.8865e-6	-4.8401e-6	6.8778e-6	-2.3610

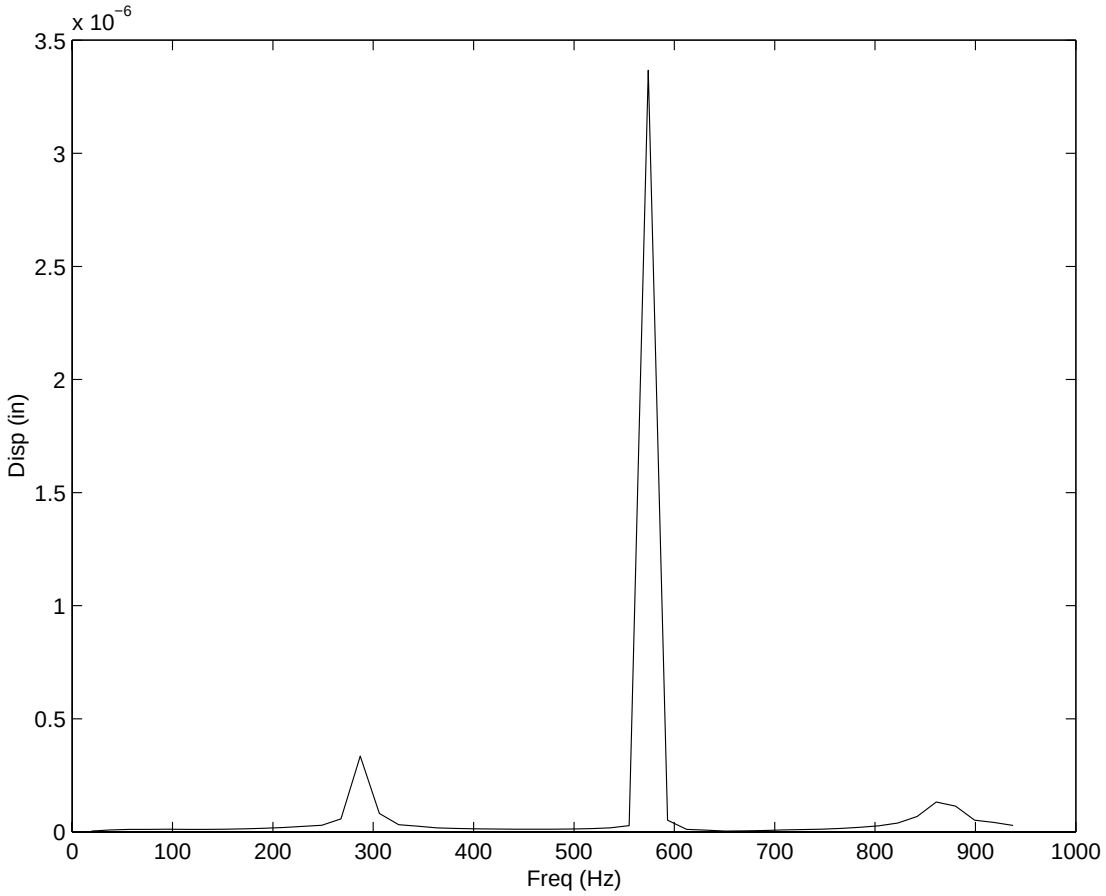


Figure 9.8: Plot showing the response near the first multiple of mesh frequency for the radial component of deformation of the pinion1 bearing (transient response is in column no.2 of the PINION1BRGRES.DAT file)

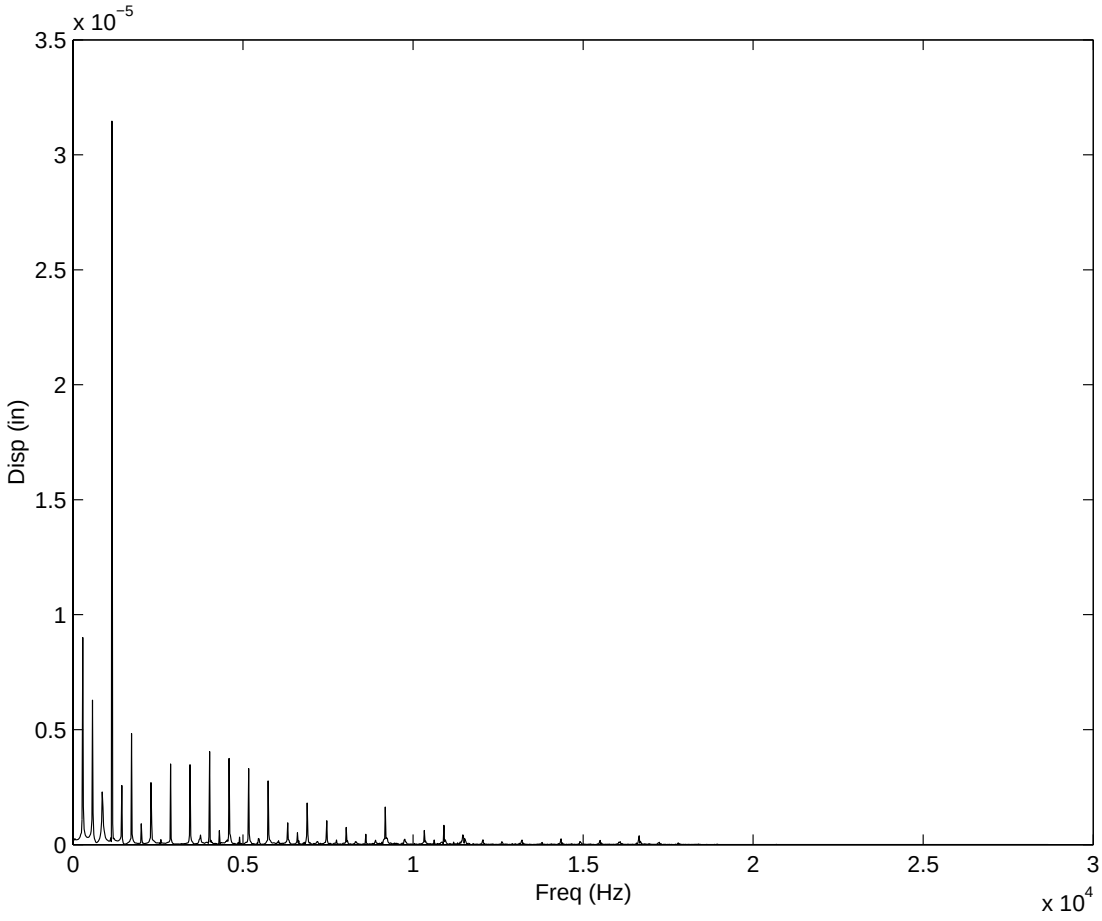


Figure 9.9: Plot showing the tangential component frequency response of pinion1 in the planetary system in the dynamic state (transient response is in column no.3 of the PINION1BRGRES.DAT file)

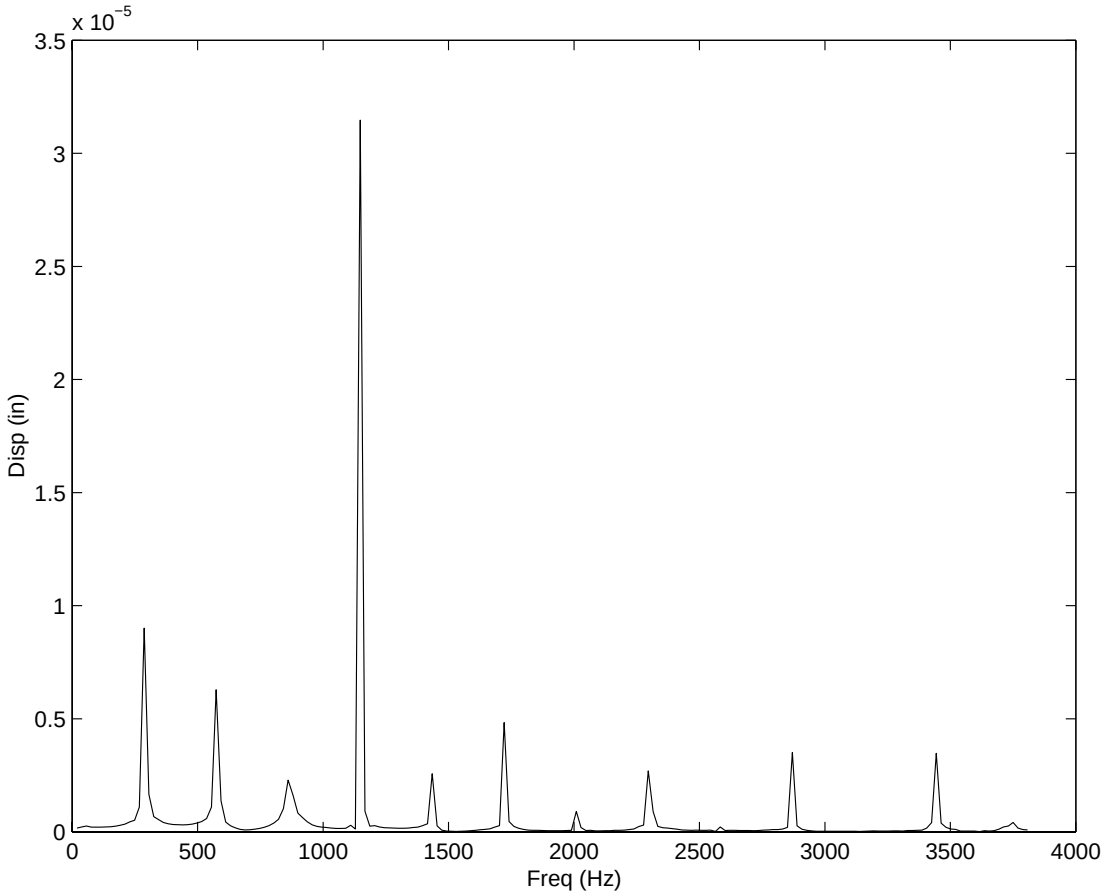


Figure 9.10: Plot showing a narrow range of tangential component frequency response of pinion1 in the planetary system in the dynamic state (transient response is in column no.2 of the PINION1BRGRES.DAT file)

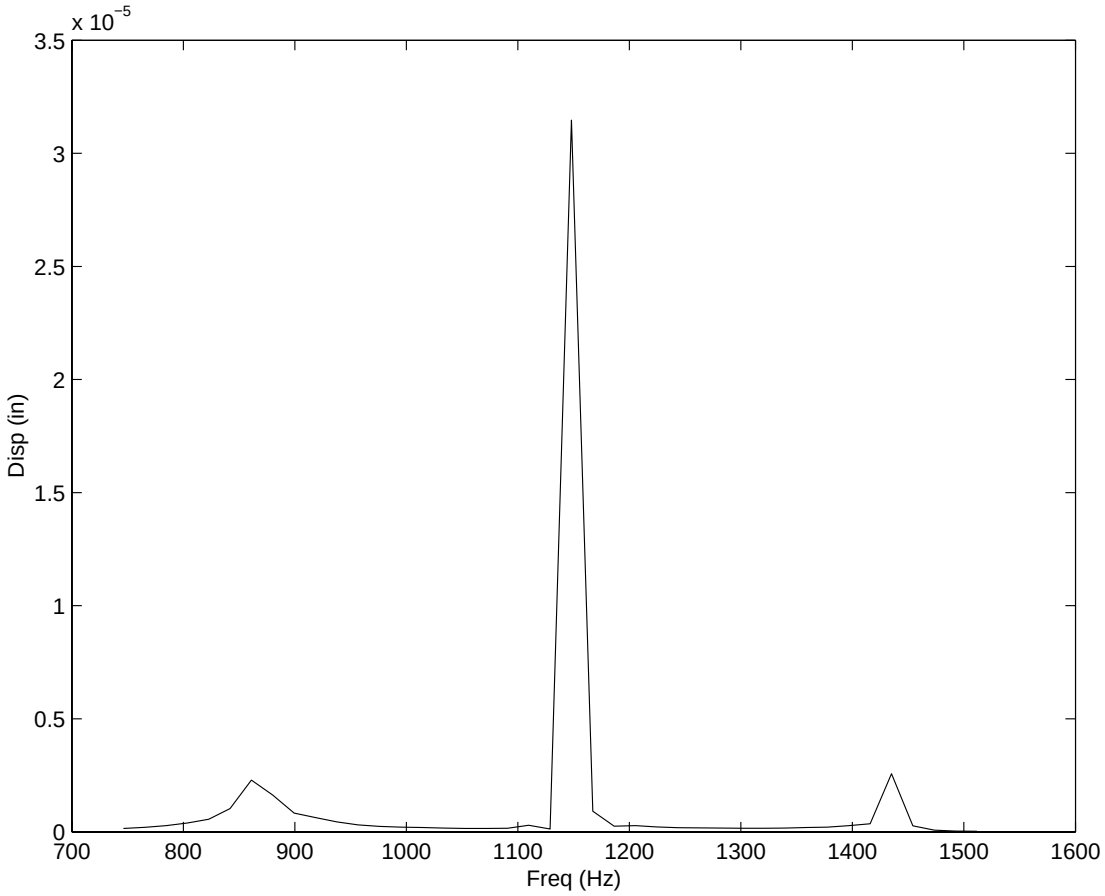


Figure 9.11: Plot showing the response near the second multiple of the mesh frequency for the tangential component of deformation of the pinion1 bearing (transient response is in column no.2 of the PINION1BRGRES.DAT file)

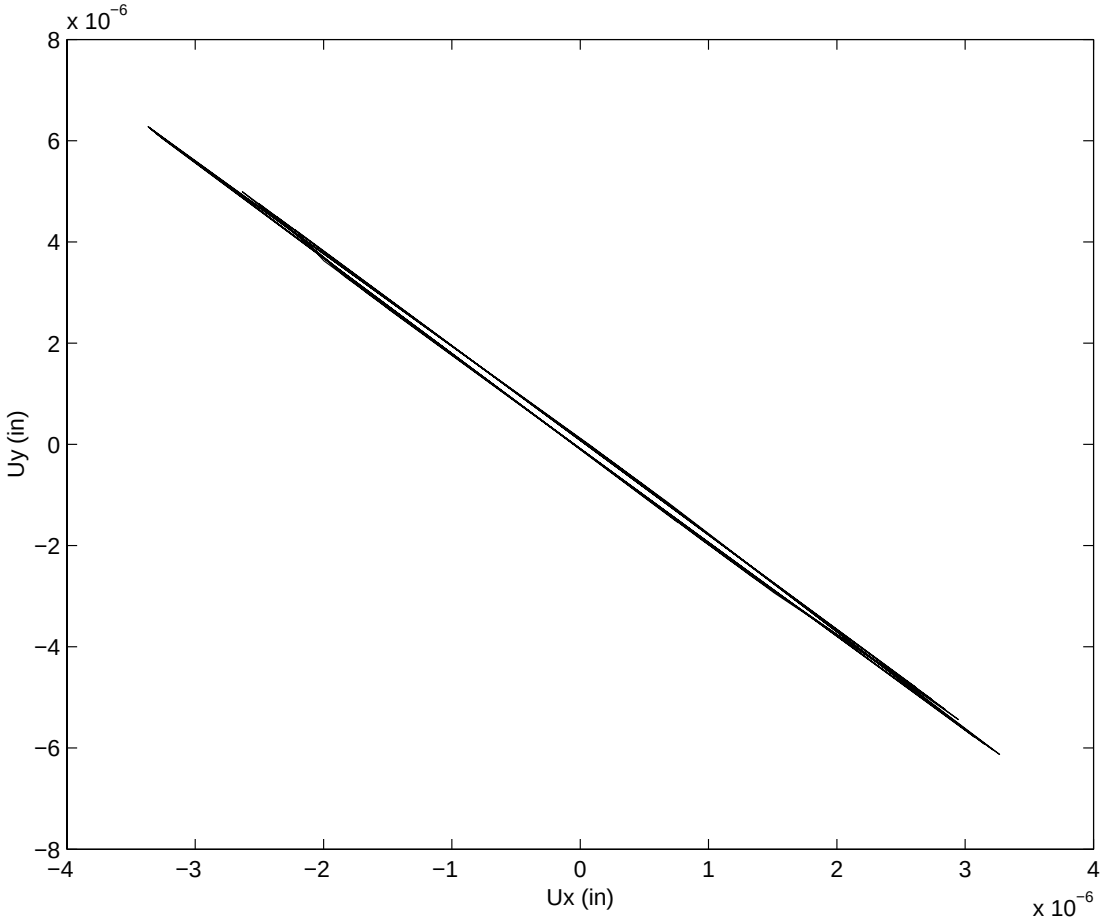


Figure 9.12: Plot showing the mode shape at pinion1 in the planetary system in the dynamic state at 574.05Hz

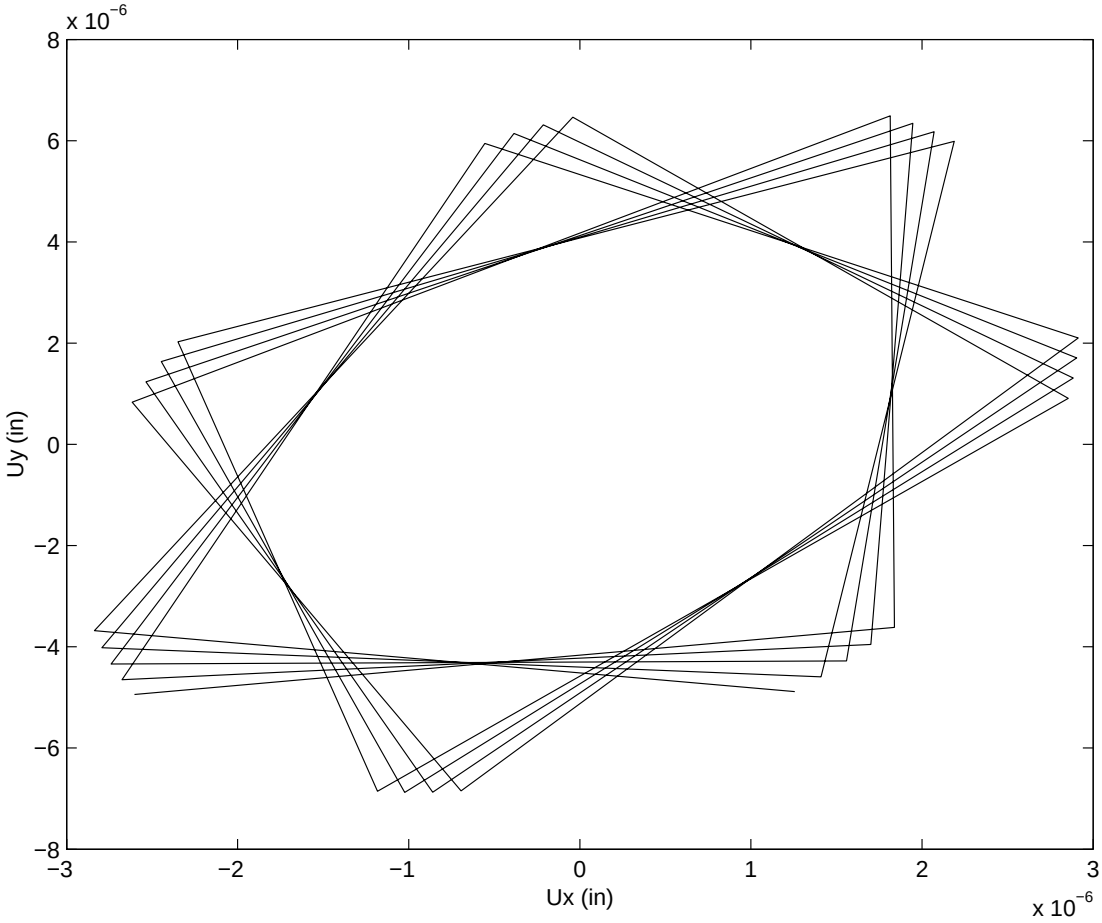


Figure 9.13: Plot showing the mode shape at the sun gear in the planetary system in the dynamic state at 574.05Hz

Appendix A

Tooth Mesh Templates

The finite element meshes in the *Planetary2D* package are created with very little input from the user. The user does not need to provide any of the node numbering and element connectivity information to the model generator. This information is read by the program from pre-existing files called 'template' files.

Figures A.1 through A.4 show the element connectivity and element numbering scheme used in the four standard templates. The orientation of the element coordinate system is indicated by the notch in one of the corners of each element. The range of the surface profile coordinate s for the two contact surfaces is also shown.

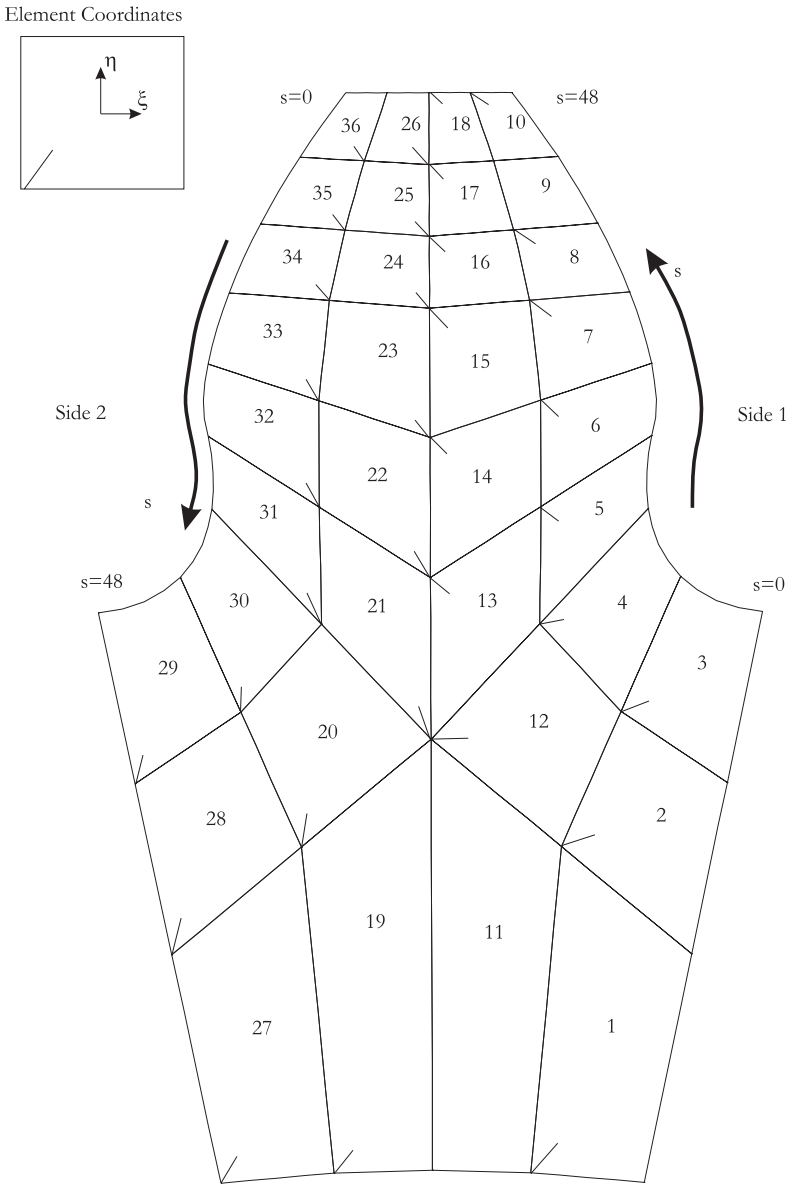


Figure A.1: The MEDIUM.TPL template file.

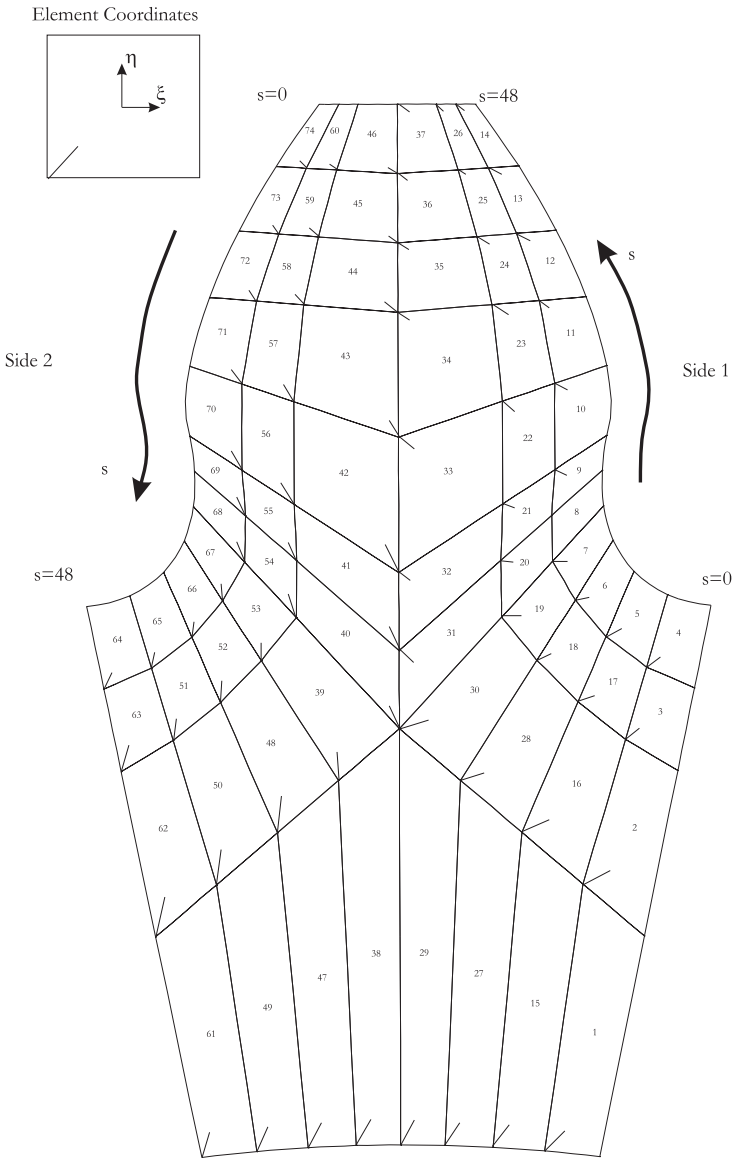


Figure A.2: The FINEROOT.TPL template file.

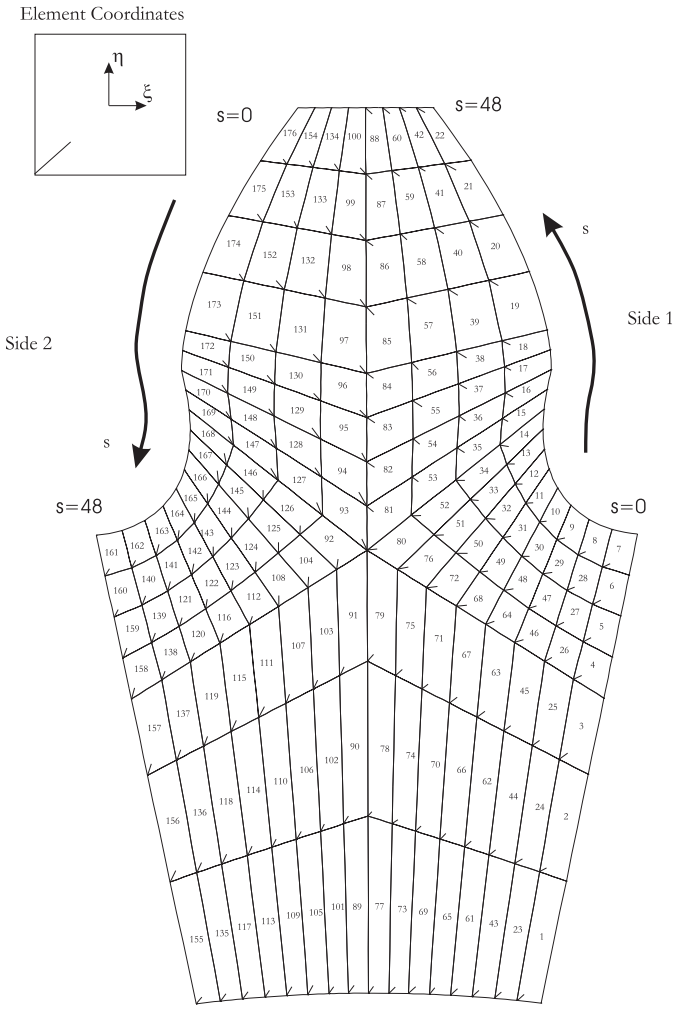


Figure A.3: The FINEST.TPL template file.

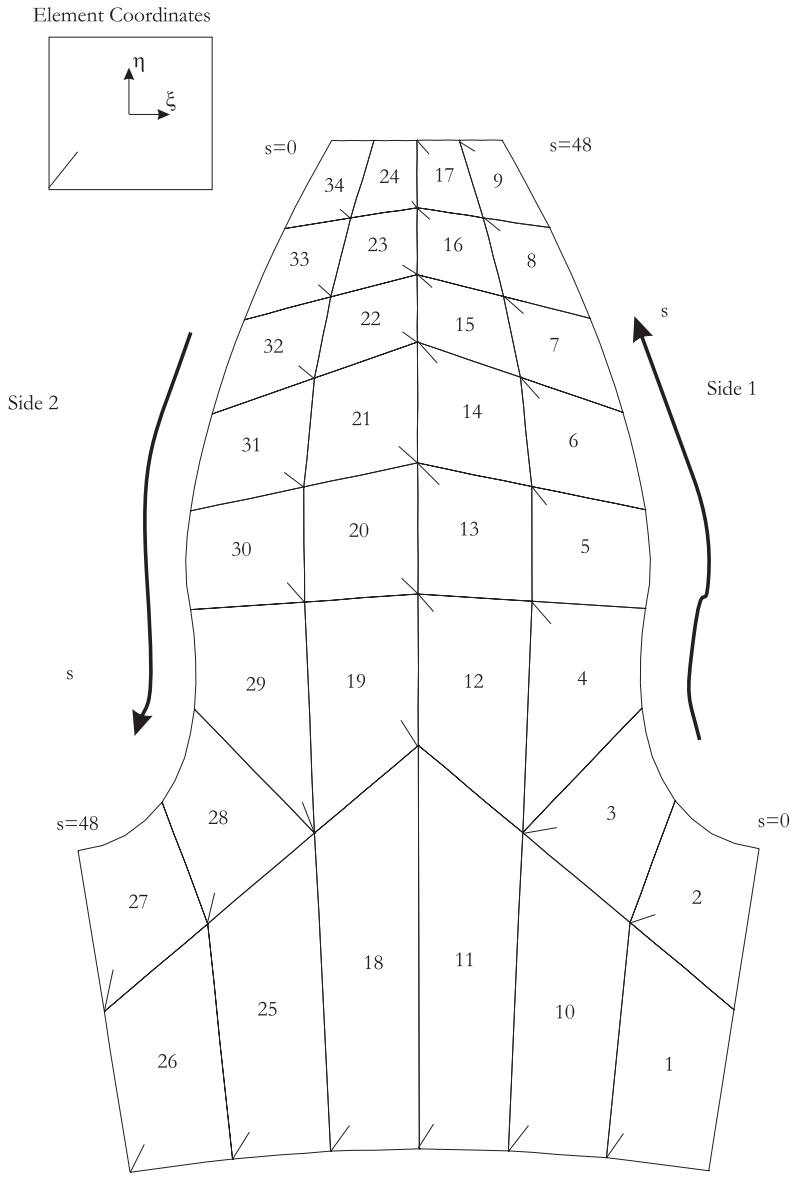


Figure A.4: The THINRIM.TPL template file.

Bibliography

[1] *Planetary Gear Train Ring Gear and Support Structure Investigation*, Mark Valco, Ph.D. Dissertation, Cleveland State University, 1992.

[2] *Gear Tooth Stress Measurements of Two Helicopter Planetary Stages*, Krantz, T. L., NASA Technical Memorandum 105651, AVSCOM Technical Report 91-C-038, 1992.

[3] A combined surface integral and finite element solution for a three-dimensional contact problem, S. Vijayakar, *International Journal for Numerical Methods in Engineering*, vol.31, pp. 525-545, 1991.

[4] *Nonlinear and dynamic programming*, G. Hadley, Addison Wesley Publishing company, 1964.

[5] *Linear programming*, George Hadley, Addison Wesley, 1962.

[6] *Linear and Combinatorial Programming*, Katta G. Murty, John Wiley, 1976 ISBN: 0-471-57370-1.

[7] Linearization of multibody frictional contact problems, S. Vijayakar, H. Busby and D. Houser, *Computers and Structures*, vol. 29, no. 4, pp. 569-576, 1987.

[8] *Natural Frequency Spectra and Vibration Modes of Planetary Gears*, Jian Lin and Robert Parker, 1998 ASME Design Engineering Technical Conference, September 1998, Atlanta Georgia.

[9] *Gear Dynamics Experiments, Part I: Characterization of Forced Response*, Blankenship and Kahraman, ASME 7^{th} International Power Transmissions and Gearing Conference, San Diego, October 1996.

[10] *Gear Dynamics Experiments, Part II: Effect of Involute Contact Ratio*, Blankenship and Kahraman, ASME 7^{th} International Power Transmissions and Gearing Conference, San Diego, October 1996.

[11] *Gear Dynamics Experiments, Part III: Effect of Involute Tip Relief*, Blankenship and Kahraman, ASME 7^{th} International Power Transmissions and Gearing Conference, San Diego, October 1996.

[12] The use of boundary elements for the determination of the geometry factor, Vijayakar and Houser, 1986 *AGMA Fall Technical Meeting*, Paper no. 86-FTM-10.

[13] Finite element analysis of quasi-prismatic structures, S. Vijayakar, H. Busby and D. Houser, *International Journal for Numerical Methods in Engineering*, vol. 24, pp. 1461-1477, 1987.

[14] Edge effects in gear tooth contact, S. Vijayakar, *ASME 7^{th} International Power Transmissions and Gearing Conference*, San Diego, October 1996.

[15] Vibration Measurements on Planetary Gears of Aircraft Turbine Engines, M. Botman, *AIAA Journal*, vol. 17, no. 5, 1980.

[16] Dynamic Tooth Loads in Epicyclic Gears, F. Cunliffe, J. D. Smith, and D.B. Welbourn, *J. Eng. Ind. Trans. ASME*, May 1974.